AFOSR - TR - 76 - 1071

AFOSR-TR-
AFATL-TR-76-55

ADA029539

THEORETICAL AND EXPERIMENTAL INVESTIGATION OF BURIED CONCRETE STRUCTURES

VOLUME II -- USER'S GUIDE TO COMPUTER CODE VANISH

W. P. VANN
J.H. SMITH

sponsored by the

AIR FORCE OFFICE OF SCIENTIFIC RESEARCH
AIR FORCE SYSTEMS COMMAND, USAF

and

AIR FORCE ARMAMENT LABORATORY
VULNERABILITY ASSESSMENTS BRANCH
AFSC, USAF, EGLIN AFB, FLORIDA

under

GRANT NO. AFOSR 74-2720

Institute for University Research-Engineering

FINAL REPORT
DECEMBER 1975

TEXAS TECH UNIVERSITY
Lubbock, Texas 79409

Qualified requestors may obtain additional copies from the Defense Documentation Center, all others should apply to the National Technical Information Service.

<u>Conditions of Reproduction</u>

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

REPORT DOCUMENTATION PAGE		READ INSTRUCTIONS BEFORE COMPLETING FORM
1. REPORT NUMBER AFOSR - TR - 76 - 1071	2. GOVT ACCESSION NO.	3. RECIPIENT'S CATALOG NUMBER
4. TITLE (and Subtitle) THEORETICAL AND EXPERIMENTAL INVESTIGATION OF BURIED CONCRETE STRUCTURES - VOLUME II, USER'S GUIDE TO COMPUTER CODE VANISH		5. TYPE OF REPORT & PERIOD COVERED FINAL 1 April 1974 — 31 Aug 1975
		6. PERFORMING ORG. REPORT NUMBER
7. AUTHOR(s) W. P. VANN, J. H. SMITH		8. CONTRACT OR GRANT NUMBER(s) AFOSR 74-2720
9. PERFORMING ORGANIZATION NAME AND ADDRESS TEXAS TECH UNIVERSITY, DEPARTMENT OF CIVIL ENGINEERING, LUBBOCK, TEXAS 79409		10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS 681307, 9782-05, 61102F
11. CONTROLLING OFFICE NAME AND ADDRESS AIR FORCE OFFICE OF SCIENTIFIC RESEARCH/NA, BUILDING 410, BOLLING AIR FORCE BASE, D C 20332		12. REPORT DATE Dec 1975
		13. NUMBER OF PAGES 58
14. MONITORING AGENCY NAME & ADDRESS(if different from Controlling Office)		15. SECURITY CLASS. (of this report) UNCLASSIFIED
		15a. DECLASSIFICATION/DOWNGRADING SCHEDULE

6 9 10 11 12 66p. 15 AF-AFOSR-2720-74

16. DISTRIBUTION STATEMENT (of this Report)

Approved for public release; distribution unlimited.

16 AF-9782 17 9782-05

17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)

18 AFOSR TR-76-1071, TR-76-Vol-1

18. SUPPLEMENTARY NOTES

19. KEY WORDS (Continue on reverse side if necessary and identify by block number)

STRUCTURES	NONLINEAR
DYNAMICS	CONCRETE
FINITE ELEMENTS	BURIED ARCHES
ANALYSIS	COMPUTER CODE
EXPERIMENTAL	USER'S GUIDE

20. ABSTRACT (Continue on reverse side if necessary and identify by block number)

This volume provides details concerning the digital computer code, VANISH, which was developed for analyzing the Static And Dynamic behavior of Nonlinear Interacting Soil Structures and the volume is intended to serve as a user's guide to VANISH. In particular, the code has been designed to be efficient and reliable in determining the response of a soil-covered circular arch used as a aircraft shelter. However, it should also be useful in analyzing a variety of other planar structures. Volume I of this report, entitled "Summary of Analysis and Experiment" gives the fundamental concepts underlying the VANISH code.

Nevertheless, most of the knowledge needed merely to use the VANISH code is available from the present volume, and requires only a general understanding of how the code actually solves a given problem.

DEPARTMENT OF CIVIL ENGINEERING

TEXAS TECH UNIVERSITY

"Theoretical and Experimental Investigation of Buried Concrete Structures"

Volume II - User's Guide to Computer Code VANISH

by

W.P. Vann and J.[illegible]. Smith

Associate Professors
and
Principal Investigators

Final Technical Report on Grant No.
AFOSR 74-2720 from the Air Force
Office of Scientific Research

Lubbock, Texas

December 1975

FOREWORD

The Research was sponsored by the Air Force Office of Scientific Research, Air Force Systems Command, USAF, under Grant No. AFOSR-74-2720. Funds were provided by the Air Force Armament Laboratory, Armament Development and Test Center (ADTC), Eglin Air Force Base, Florida under Obligation Authorities 74-Y-7 and 75-V-11. Mr. Phillip T. Nash (DLYV) was the technical program manager for the Armament Laboratory.

The effort was begun on 1 April 1974 and was completed on 31 August 1975.

CONTENTS - VOLUME II

User's Guide to Computer Program VANISH

I. INTRODUCTION

This volume provides details concerning the digital computer code, VANISH, which was developed for the Vulnerability Analysis of Nonlinear Interacting Soil and Hardened Structures. In particular, the code as been designed to be efficient and reliable in determining the response of a soil-covered circular arch used as an aircraft shelter. However, it should also be useful in analyzing a variety of other planar structures. Since the arch of immediate interest is composed of precast reinforced concrete ribs, and the response of the system is of interest all the way to collapse, special measures have been taken to account for nonlinear behavior of concrete members, and of nonlinear behavior of the soil.

Volume I of this report, entitled "Analysis and Experiment" gives the fundamental concepts underlying the VANISH code. In particular, Chapter II of Volume I gives description of the modeling techniques to be used for various structures, derivations of the stiffness and mass properties of the three types of finite elements (beam, bar and constant strain triangle elements) used in the code, explains how the code handles camping and applied forces (including blast pressures), and presents the methods used for incremental static loading and for dynamic loading. The reader will find understanding of and ready reference to Chapter II of Volume I essential to understanding of the latter portions of the present volume, which document the operation of the code.

Nevertheless, most of the knowledge needed merely to use the VANISH code is available from the present volume, and requires only a general understanding of how the code actually solves a given problem.

The essential steps in preparing to use the code to analyze a structure are: 1) to idealize the actual structure by means of straight beam elements and/or triangular soil elements, 2) to choose lines of possible soil cracking between triangular elements,

if such nonlinear behavior of the soil is to be considered, and introduce short bar elements across these potential crack lines, 3) to number all of the nodes and elements (by element type), 4) to evaluate the stiffness and mass properties of each element, 5) to choose the damping coefficients to be specified for one or two nodes, 6) to decide what loads are to be applied (these must be concentrated at the nodes), and 7) to decide what results are to be printed out. The steps in describing this data to the computer code by means of punched cards are presented in Chapter II of this volume.

The remaining chapters in this volume give detailed information about the computer code itself, so that potential problems in its operation and extensions of its capabilities may be handled by future users. Chapter III gives the basic function and Chapter IV presents a flowchart of each subroutine and shows how all of the subroutines are related to one another (mostly through the MAIN program). Chapter V defines the important variable names used and identifies the subroutines and COMMON blocks where they occur.

II. INPUT/OUTPUT INSTRUCTIONS

A. General

The first step in the analysis of a structure by the finite element method is to divide the structure into finite elements connected at points called nodes. The use of this code is limited to plane stress problems employing straight beam and bar elements and plane triangular elements. A suitable x-y coordinate system should be selected and the coordinates of the nodes determined. The coordinates of the nodes, material properties of the elements, boundary conditions, damping characteristics and loads define the problem to be solved. These data are to be input by means of punched cards.

B. Input Cards to Define the Structure

1. Title Card

The first card of the data set for any individual problem must have a problem number in columns 1 to 5. This is to enable a number of problems to be solved one after the other. A blank card must be used at the end of the complete data set to specify the end of all the problems. The title card also allows a title to be given to each individual problem treated, and this title is printed as a heading on the output.

FORMAT (I5, 15A4)

Columns	Data
1 - 5	Problem number.
6 - 65	Title of the problem.

2. Standard Options Card

Standard options are required in all problems, whereas non-linear options (following card) are required only for nonlinear problems.

FORMAT (6I5)

Columns	Data
1 - 5	Parameter specifying the type of problem. If 1, static analysis. If 0 (or blank), dynamic analysis.
6 - 10	Parameter specifying linear or nonlinear analysis. If 1, linear analysis. If 0 (or blank), nonlinear analysis.
11 - 15	Parameter specifying whether the stresses are to be determined. If 1, the displacements only are determined. If 0 (or blank), the displacements and stresses are determined.
16 - 20	Parameter controlling printout of results. If 1, the displacements (and stresses if calculated) at specified nodes only are printed. If 0 (or blank), displacements, velocities, accelerations (and stresses if calculated) are printed for all the nodes.
21 - 25	Parameter specifying whether dynamic forces are read in or generated internally by subroutine FORGEN. If 1, FORGEN is used. If 0 or blank forces are read in.
26 - 30	Parameter specifying inclusion of dead loads. If 1, dead loads area neglected. If 0, dead loads are included.

3. Nonlinear Options Card

This card is omitted if the analysis is linear.

FORMAT (6I5)

Columns	Data
1 - 5	Parameter specifying which elements (beams, bars, or both) are nonlinear. If 0, only bars are nonlinear. If 1, only beams are nonlinear. If 2, both are nonlinear.
6 - 10	Parameter specifying type of beam material in axial deformation. If 0, general homogeneous material with symmetric yield levels. If 1, reinforced concrete (separate steel and concrete).
11 - 15	Parameter specifying how incremental static loads are prescribed. If 0, increments are set internally. If 1, increments are read in.

16 - 20	Parameter specifying print-out at incremental static load levels. If 0, no print-out at each load. If 1, standard output at each load.

4. Problem Size Card

FORMAT (5I5)

Columns	Data
1 - 5	Number of nodal points (150 maximum*)
6 - 10	Number of beam elements (30 maximum*)
11 - 15	Number of bar elements (100 maximum*)
16 - 20	Number of triangular elements (200 maximum*)
21 - 25	Number of nodal points having specified boundary conditions (33 maximum*)

(*for dimensions in present code version)

5. Nodal Point Coordinate Cards

One card is required for each nodal point in numerical sequence.

FORMAT (I5, 5X, 2F10.0)

Columns	Data
1 - 5	Nodal point number.
6 - 10	Blank
11 - 20	X-Coordinate (units-feet)
21 - 30	Y-Coordinate (units-feet)

6. Beam Element Data Cards - Linear Behavior

These cards are omitted if there are no beam elements in the finite element system. One card is required for each beam element in numerical sequence.

FORMAT (3I5, 5X, 4F10.0)

Columns	Data
1 - 5	Beam element number.
6 - 10	Nodal point number of one end of beam element.
11 - 15	Nodal point number of the other end of the beam element.
16 - 20	Blank
21 - 30	Area of cross-section (in^2)
31 - 40	Moment of inertial of the cross-section (in^4)
41 - 50	Modulus of elasticity of the beam material (psi)
51 - 60	Unit weight of the beam material (lb/ft^3)

7. Beam Element Data Cards - Nonlinear Behavior

These cards are omitted if the beam elements remain linear, but are needed in addition to the linear data cards for nonlinear beams.

a) Nonlinear Axial Deformation Properties

(One card is required for each beam element in numerical sequence. Columns 41-80 may be left blank for homogeneous beams).

FORMAT (I5, 5X, 7F10.0)

Columns	Data
1 - 5	Beam element number
6 - 10	Blank
11 - 20	Yield stress (psi) of homogeneous beam or steel of R/C beam
21 - 30	Rupture strain (in/in) of homogeneous beam or steel of R/C beam
31 - 40	Rupture stress (psi) of homogeneous beam or steel of R/C beam
41 - 50	Area of steel (in^2) of R/C beam
51 - 60	Compressive yield stress (psi) of concrete R/C beam (f'_c).
61 - 70	Rupture strain in compression of concrete of R/C beam
71 - 80	Rupture stress (psi) in compression of concrete of R/C beam

b) Nonlinear Bending Deformation Properties

(One card is required for each beam element in numerical sequence).

FORMAT (I5, 5X, 7F10.0)

Columns	Data
1 - 5	Beam element number
6 - 10	Blank
11 - 20	Positive yield moment of each rotary spring (ft - kips)
21 - 30	Absolute value of negative yield moment of each rotary spring (ft - kips)
31 - 40	Positive rupture rotation of each rotary spring (radians)

41 - 50	Absolute volume of rupture rotation of each rotary spring (radians)
51 - 60	Positive rupture moment of each rotary spring (ft - kips)

8. Bar Element Data Cards

These cards are omitted if there are no bar elements in the finite element system. One card is required for each bar element in numerical sequence.

FORMAT (3I5, 5X, 3F10.0, I10, F10.0)

Columns	Data
1 - 5	Bar element number
6 - 10	Nodal point number of one end of the bar element
11 - 15	Nodal point number of the other end of the bar element
16 - 20	Blank
21 - 30	Area of crossection (in^2)
31 - 40	Modulus of elasticity of the bar material (psi)
41 - 50	Unit weight of the bar material (lb/ft^3)
51 - 60	Parameter specifying how yielded (tensile) stiffness is prescribed. If = 1, read in by next parameter. If = 0, specify internally as 10^{-8}.
61 - 70	Yielded (tensile) stiffness as ratio to compressive stiffness, EA/L.

9. Triangular Element Data Cards

These cards are omitted if the number of triangular elements in the problem is zero. One card is required for each triangular element in numerical sequence.

FORMAT (4I5, 4F10.0)

Columns	Data
1 - 5	Triangular element number
6 - 10	*Node number of 1st node of the triangular element
11 - 15	*Node number of the 2nd node
16 - 20	*Node number of the 3rd node

*The three node numbers of each triangular element must be in counterclockwise sequence.

21 - 30	Thickness of the element (ft)
31 - 40	Modulus of elasticity of the material (psi)
41 - 50	Poisson's ratio of the material
51 - 60	Unit weight of the material (lb/ft^3)

10. Boundary Condition Cards

One card is required for each nodal point where displacement boundary conditions are prescribed. The code is written such that all prescribed boundary conditions must have zero values. These cards need not be in any numerical sequence. However, the total number of cards must be equal to the number specified on the problem size card.

FORMAT (4.I5)

Columns	Data
1 - 5	Nodal point number at which displacement boundary conditions are specified.
6 - 10	1, if the u displacement (displacement in the x direction) is zero. Otherwise blank.
11 - 15	1, if the v displacement (displacement in the y direction) is zero. Otherwise blank.
16 - 20	1, if the rotation is zero. Otherwise blank.

11. Dynamic Analysis Parameters Card

FORMAT (3F10.0, I10)

Columns	Data
1 - 10	Coefficient by which to multiply the stiffness matrix to get the "relative" part of the damping matrix.
11 - 20	Coefficient by which to multiply the mass matrix to get the "absolute" part of the damping matrix.
21 - 30	Magnitude of the time step (seconds).
31 - 40	Maximum number of time steps to which the analysis is to be carried out.

C. Input Cards to Control the Output

The following cards may be omitted if the displacements, velocities, accelerations and stresses are desired to be printed out for

all nodal points. Columns 15 - 20 in the "options card" (following the title card) must be blank in this case. If the output information is desired only at certain (special) nodal points, these nodal point numbers are input using the following cards. Column 20 in the "options card" must contain 1 in this case.

1. Number of Special Nodes Card

FORMAT (I5)

Columns	Data
1 - 5	Total number of nodal points whose displacements, stresses, etc. are desired to be printed out.

2. Cards Specifying Special Node Numbers

FORMAT

Columns	Data
1 - 5	
6 - 10	Nodal point numbers whose displacements, velocities, etc., are to be printed out.
- - - -	
- - - -	
61 - 65	
66 - 70	

Each card contains 14 numbers. Use as many cards as are necessary to specify all the node numbers.

3. Numbers of Special Elements Card

FORMAT (5I5)

Columns	Data
1 - 5	Total number of beam elements for which member forces are to be printed out.
6 - 10	Total number of bar elements for which for which forces are to be printed out.
11 - 15	Total number of triangular elements for which stresses are to be printed out.

4. Cards Specifying Special Element Numbers

FORMAT (14I5)

Columns	Data
1 - 5	
6 - 10	Element numbers whose forces (beams and/or
- - - -	bars) or stresses (triangles) are to be
- - - -	printed out; all beams first, then all bars,
61 - 65	then all triangles.
66 - 70	

D. Input Cards for Applied Loads

One set of cards must be included for each time step. In each set, one card is required for each nodal point where external loads are applied. The cards need not be in any particular numerical sequence. However, the last card in a set must be blank indicating the end of a set of loads for a particular time step.

FORMAT (I5, 5X, 3F10.0)

Columns	Data
1 - 5	Node number where the external loads are applied.
6 - 10	Blank
11 - 20	X-Component of the Load (kips), positive upward
21 - 30	Y-Component of the load (kips), positive to right
31 - 40	Moment (ft kips), positive counterclockwise

Note that in the present version of the code, there is no provision to input distributed loads (other than the dead weight which is automatically calculated using the unit weights of the materials).

All the preceding cards specify one problem. Data for a number of problems may follow one after the other. The end of the data for all problems is indicated by introducing a blank card after the complete data.

E. Output

There is only one type of output, namely printed output. The following information is printed by the program for all problems:

a. Input data. All the input data except the loads are printed before the execution of the problem.

b. Error messages. The code checks the dimensions provided against the size of the problem and prints out error messages if the size exceeds the dimensions provided.

The following information is printed for each load increment, if desired, and for the total load for static loading:

a. Displacements of each node or designated nodes.

b. Axial force, shear, and moment at each end of each beam element or designated elements.

c. Axial force in each bar element, or designated elements.

d. Averaged stresses at the nodal points of each triangular element, or designated elements.

The following information is printed for each time step:

a. Nodal displacements and velocities of each node, or designated nodes.

b. Forces or stresses in each element, or in designated elements, if called for (if NOUT = 1).

III. MAIN PROGRAM AND SUBROUTINES

A. Main Program

The MAIN program is a control program. As shown in the accompanying flowchart, its major steps are:

1. Read and print the problem number, title and options (static or dynamic analysis, linear or nonlinear analysis).

2. Call subroutine SETUP in which most of the data are read.

3. Call subroutine ASMBLE which assembles the stiffness and mass matrices.

4. If the analysis is static, call BCOND to incorporate the boundary conditions into the stiffness matrix, call DECOMP to decompose the stiffness matrix, add dead and applied loads if appropriate, and call STSOL to solve for full static displacements.

5. If the analysis is static but may be nonlinear, call RESET to check for yielding, and call STNON to perform an incremental solution, if there is yielding.

6. If the analysis is dynamic, perform a static solution first if dead loads are considered, then perform a dynamic analysis, using SOLVE for the incremental response and RESET to check for yielding and modify the stiffness matrix and restoring force vector at each time step. Dynamic loads may either be read in or calculated by FORGEN.

The subroutines called by MAIN are:

SETUP
ASMBLE
BCOND
DECOMP
STSOL
RESET
STNON
FORGEN
SOLVE

B. Subroutine SETUP

This is the primary subroutine for reading in the data, although some data are read in MAIN. The major steps in this subroutine are:

1. Read and print out number of nodes, number of elements and other quantities specifying the size of the problem.
2. Read and print out coordinates of the nodes.
3. Read and print out data for the beam elements.
4. Read and print out data for the bar elements.
5. Read and print out data for the triangular elements.
6. Read and print out specified displacement boundary conditions.
7. Read and print parameters for dynamic analysis.
8. Read node numbers and elements whose responses are to be output.
9. Read and print loads for static analysis (or the initial loads for dynamic analysis).
10. Calculate size of matrices.
11. Check dimensions.

SETUP does not call any other subroutines.

C. Subroutine ASMBLE

This subroutine is for assembling the stiffness and mass matrices. The modified matrices for the dynamic analysis (based on the recursive form of Newmark's β method) are also developed. The loads due to the dead weight are calculated in this subroutine. The major steps involved in this subroutine are:

1. Initialize the stiffness and mass matrices.
2. Assemble the stiffness and matrices for the beam elements by calling subroutine BEAM2.
3. Assemble the stiffness and mass matrices for the bar elements by calling subroutine BAR.
4. Assemble the stiffness and mass matrices for the triangular elements by calling subroutine CST.

5. Store the stiffness matrix on tapes 1 and 3.
6. Calculate the loads due to the deadweight.

The subroutines called by ASMBLE are:

BEAM2
BAR
CST

D. Subroutine BCOND

This subroutine incorporates the boundary conditions into the stiffness matrix, that is, it sets the off-diagonal terms equal to zero in the row and column of each boundary condition and it sets the diagonal term equal to one. It does not call any other subroutine.

E. Subroutine DECOMP

This subroutine reduces a symmetric banded matrix into an upper triangular matrix by Gauss elimination. There is no provision for pivoting in this subroutine since the matrices involved in structural analysis are generally positive definite and diagonally dominant. No other subroutines are called.

F. Subroutine STSOL

This subroutine is used in the static analysis only. The solution for the displacement vector is obtained using this subroutine, which calls subroutine BSUB for this purpose. The steps involved are:

1. Print the force vector.
2. Incorporate the displacement boundary conditions in the force vector.
3. Convert the forces and moments to inch and pound units.
4. Call subroutine BSUB to solve for the displacements.
5. Print the displacements.

This subroutine calls only the subroutine BSUB.

G. Subroutine RESET

This subroutine calculates the forces and stresses in individual structural elements, prints these data if desired, checks for changes in stiffness or restoring force due to yielding, unyielding or rupture, and modifies the stiffness matrix and the restoring force vector as needed for the next time or load increment. The major steps are:

1. Calculate each bar force, print if desired, calculate each bar stiffness for next step, and change global stiffness if necessary.

2. Calculate each beam's member deformations and forces, print forces if desired, call MODBM to check for stiffness, energy, or restoring forces changes and change global stiffness if necessary.

3. Calculate each triangle's average stresses and print, if necessary.

Reset calls the following subroutines:

BAR
BEAMZ
MODBM
MULT2

H. Subroutine STNON

This subroutine sets up and controls the incremental static solution needed when there is nonlinear behavior under static loading. For each step it calculates the load increment, calls STSOL to find the increment deflections, adds the incremental loads and deflections to the running totals, prints the results if desired, and calls RESET to check for (further) nonlinear behavior.

STNON calls the following subroutines:

BCOND
DECOMP
STSOL
RESET

I. Subroutine FORGEN

This subroutine allows internal calculation of nodal forces for dynamic problems, rather than having to read them in at each time step. It is especially designed to handle the complex loading history associated with an internal blast within a soil-covered circular arch, as simplified from scale model experiments. It does not call any other subroutines. This subroutine was developed by Mr. Nash/DLYV/AFATL at Eglin AFB, Florida and its documentation is therefore beyond the scope of this report.

J. Subroutine SOLVE

SOLVE performs a step-by-step numerical integration of the differential equations of dynamic equilibrium, using Newmark's β method with $\beta = 1/4$. The steps involved are:

1. Determine the accelerations at the start of integration from the initial conditions. This step is carried out for the 1st time step only.

2. Add the dead weight to the force vector due to external loads at the end of the time step. (The force vector is input in the MAIN program for each time step).

3. Derive the modified force vector for substitution into the recursive equation of Newmark's β method.

4. Incorporate the displacement boundary conditions into the modified force vector.

5. Solve for the accelerations at the end of the time step by calling subroutine BSUB.

6. Determine the velocities and displacements at the end of the time step.

7. Store the above values as starting values for the next time step.

8. Print the desired results.

The subroutine SOLVE calls the following subroutines:

BSUB
MULT

K. Subroutine BEAMZ

This subroutine derives both the local and global stiffness matrices for a straight prismatic beam element with rotary springs at the two ends, based on the geometry of the element and its current stiffness properties. It also derives the compatibility matrix for the beam element and uses it to convert the local stiffness evaluated by subroutine BMK to a global stiffness. The subroutines called are:

BMK
MULTZ

L. Subroutine BMK

This subroutine derives the current local stiffness matrix for a beam element. The steps are:

1. Determine the axial stiffness for a general beam or for the steel in a reinforced concrete beam, EA/L.

2. Add in the concrete axial stiffness if the the beam is of reinforced concrete.

3. Determine the current bending stiffness based on the current end spring stiffnesses.

No subroutines are called by BMK.

M. Subroutine BAR

This subroutine derives the stiffness and mass matrices of a straight bar element (axial deformation only). The stiffness factors only change by proportion and do not have to be re-calculated by this subroutine during nonlinear behavior. The subroutine calls no other subroutines.

N. Subroutine CST

This subroutine derives the stiffness and mass matrices of a constant strain plane triangular element. Each node is considered to have two translational and one rotational degree of freedom, but the stiffnesses corresponding to rotation are zero. No other subroutines are called.

O. Subroutine BSUB

This subroutine solves a set of simultaneous linear equations using the upper triangular matrix obtained from subroutine DECOMP. The steps involved are:

1. Perform a forward substitution for the input vector (forces).

2. Perform a back-substitution for the solution vector (displacements or accelerations).

No other subroutines are called.

P. Subroutine MODBM

This subroutine, which is called from RESET, uses HYST to calculate the yielding behavior of a single beam element at a particular step of the solution. It considers the axial deformation, including both steel and concrete components for a reinforced concrete beam, and the bending deformation of the beam at each end, and evaluates the dissipated energy and all stiffness and restoring force changes. It calls the following subroutines:

HYST
BMK
MULTZ

Q. Subroutine HYST

This subroutine determines the changes in a general, nonsymmetric bilinear hysteretic diagram, given the current displacement, permanent set point and stiffness. The changes include possible stiffness changes due to yielding or unyielding, possible restoring force drop due to rupture, and energy changes due to any (further) yielding. The major steps are:

1. Determine if positive or negative rupture has occurred and modify the restoring force and stiffness accordingly.

2. Determine if the system remains elastic or yielded in either direction or yields for the first time, and set the stiffness keys accordingly.

3. Calculate the dissipated energy at the current displacement as an incremental part associated with the permanent set.

No other subroutines are called by HYST.

R. Subroutine MULT

This subroutine performs a matrix vector multiplication of the form $y = A\ x$ where A is a symmetric and banded matrix one half of which is stored as a singly subscripted array. No other subroutines are called.

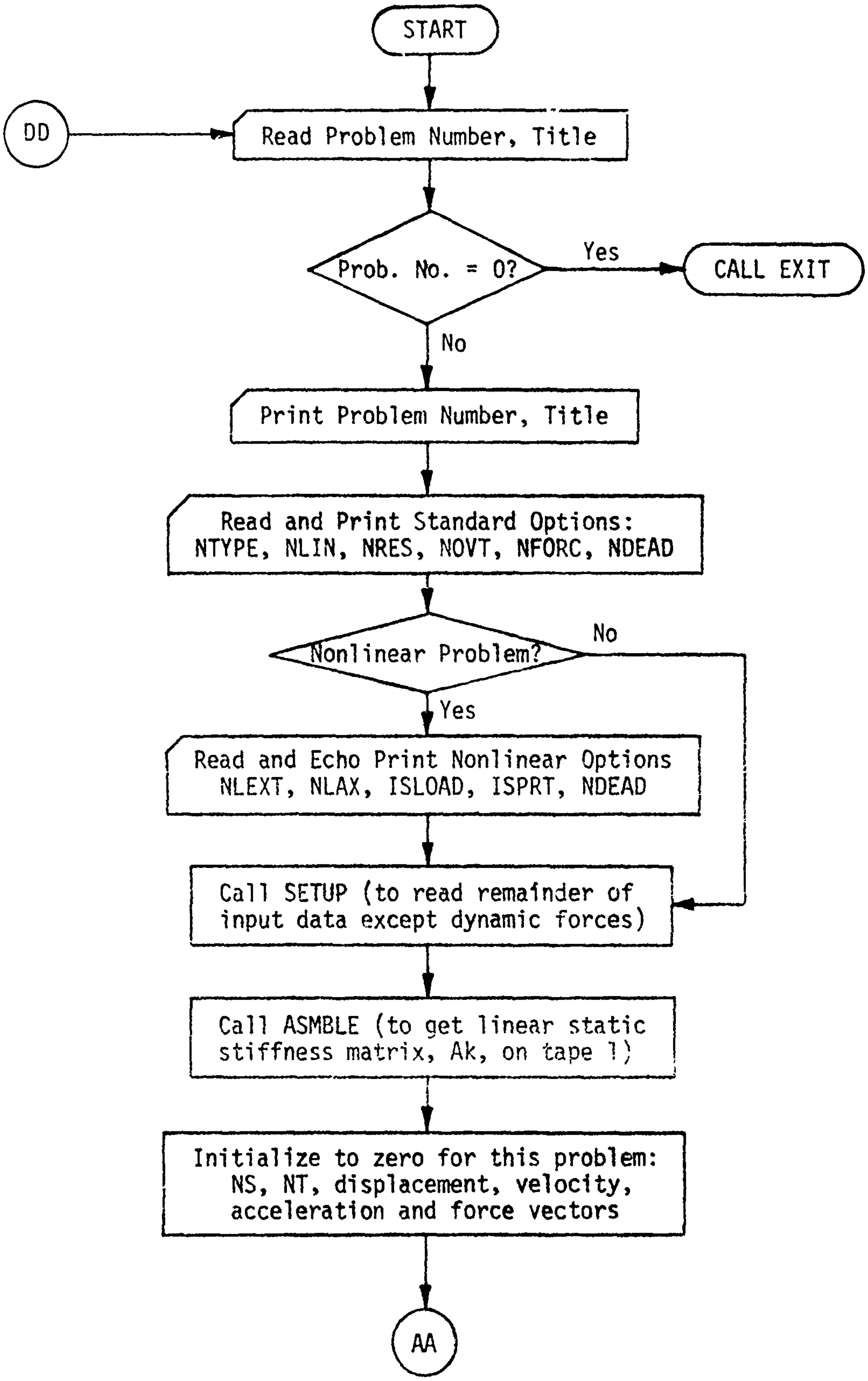
S. Subroutine MULT2

This subroutine performs a matrix multiplication of the form [A] = [B] [C], where [B] has dimension N x N1, [C] dimensions N1 x M, and [A] comes out with dimensions N x M. No other subroutines are called.

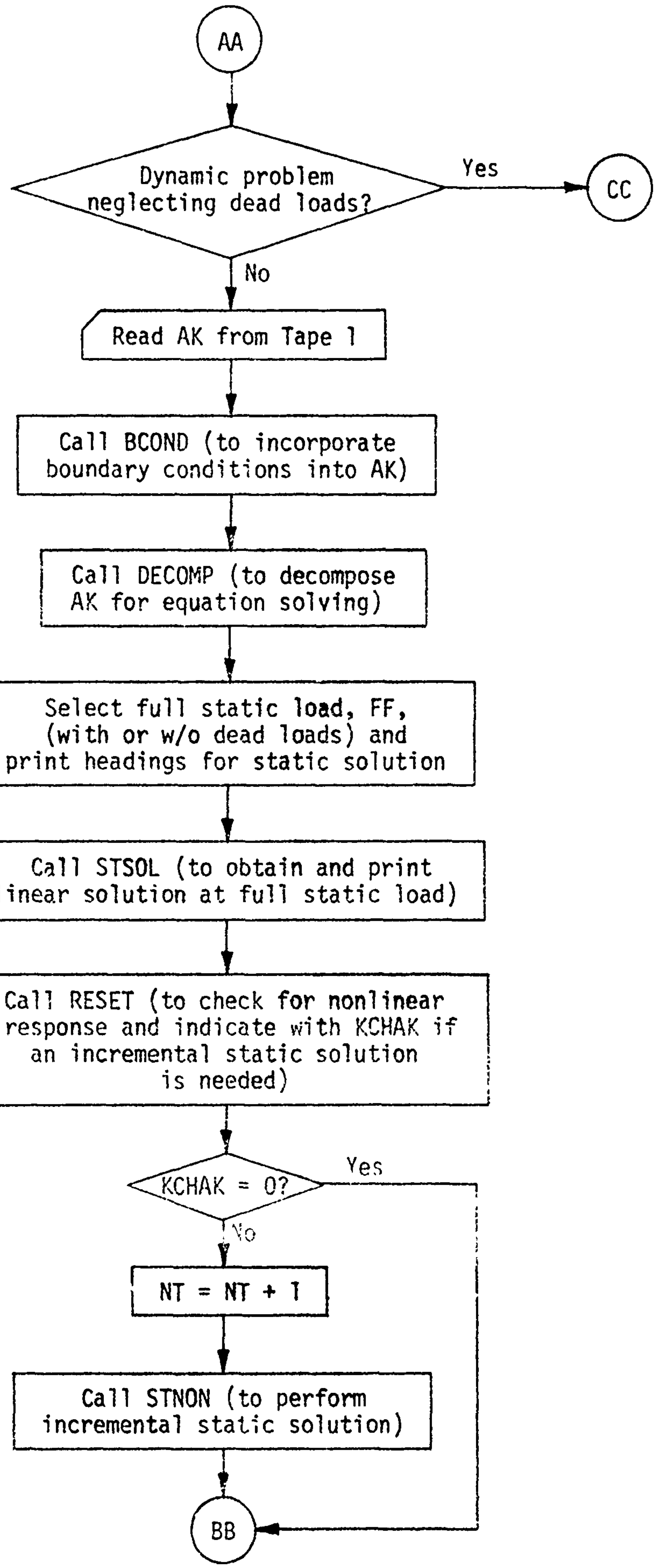
IV. FLOWCHARTS

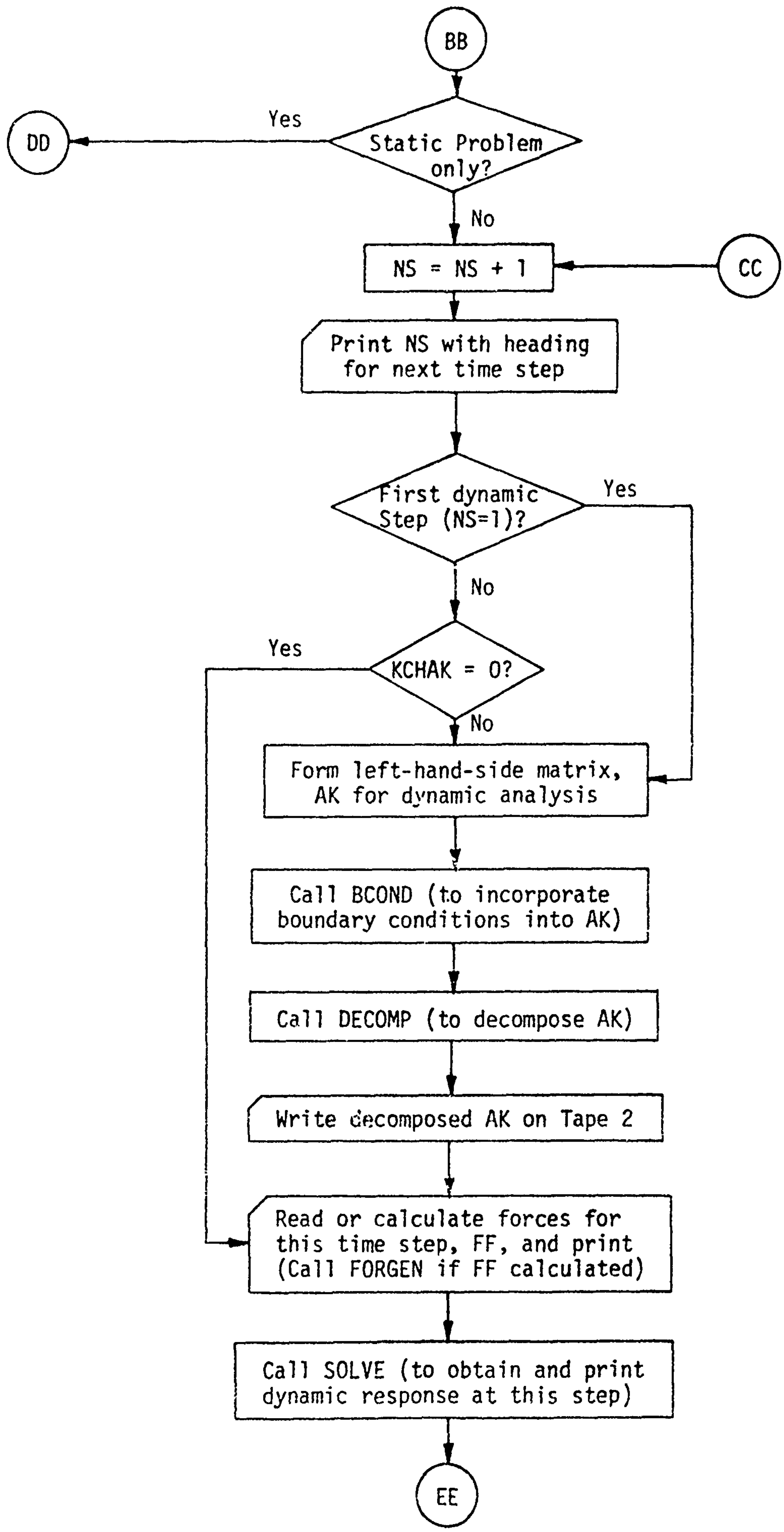
For the MAIN program and for each of the subroutines described briefly in Chapter III, a flowchart giving more details is presented in the following pages. Some of the variables used are defined in the flowcharts; others may be found in the descriptions of major variables to be found in Chapter VI.

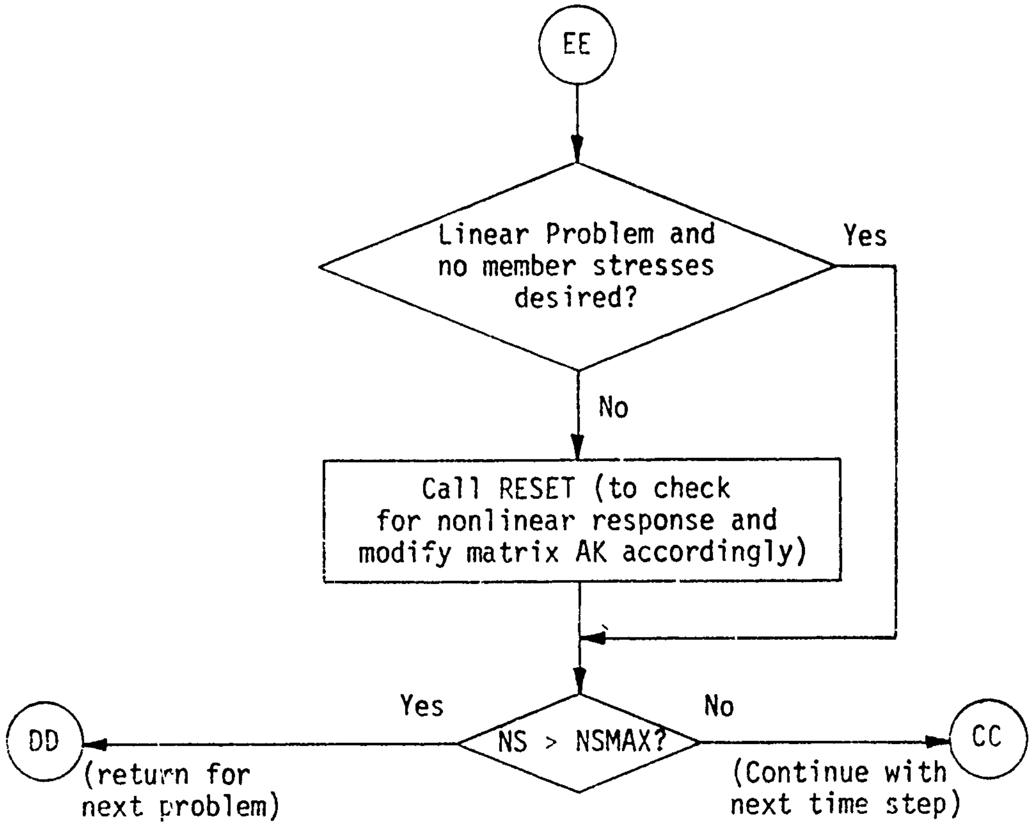
A. MAIN


START
DD
Read Problem Number, Title
Prob. No. = 0?
Yes
CALL EXIT
No
Print Problem Number, Title
Read and Print Standard Options:
NTYPE, NLIN, NRES, NOVT, NFORC, NDEAD
Nonlinear Problem?
No
Yes
Read and Echo Print Nonlinear Options
NLEXT, NLAX, ISLOAD, ISPRT, NDEAD
Call SETUP (to read remainder of input data except dynamic forces)
Call ASMBLE (to get linear static stiffness matrix, Ak, on tape 1)
Initialize to zero for this problem:
NS, NT, displacement, velocity,
acceleration and force vectors
AA

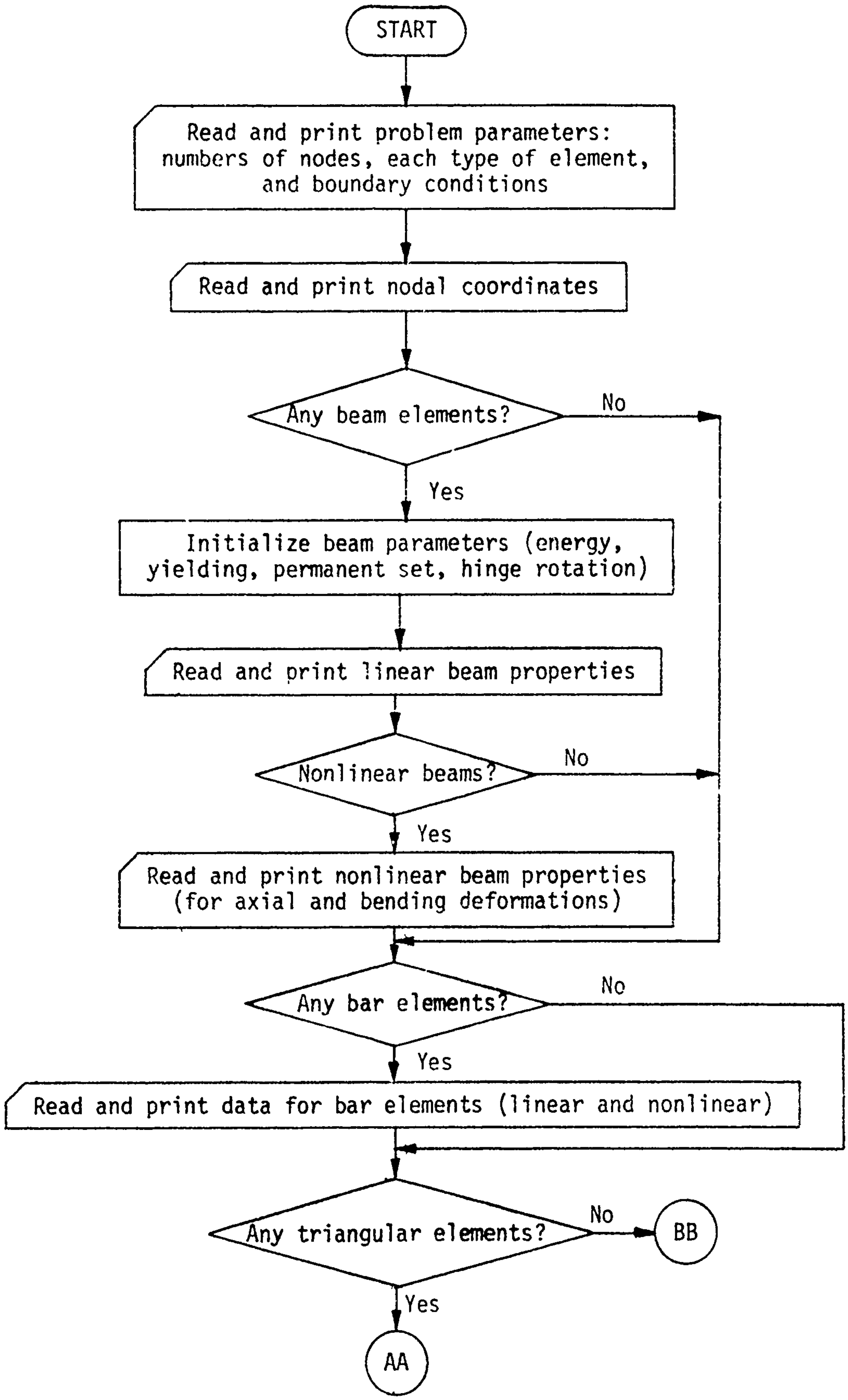
A. MAIN (Cont'd.)

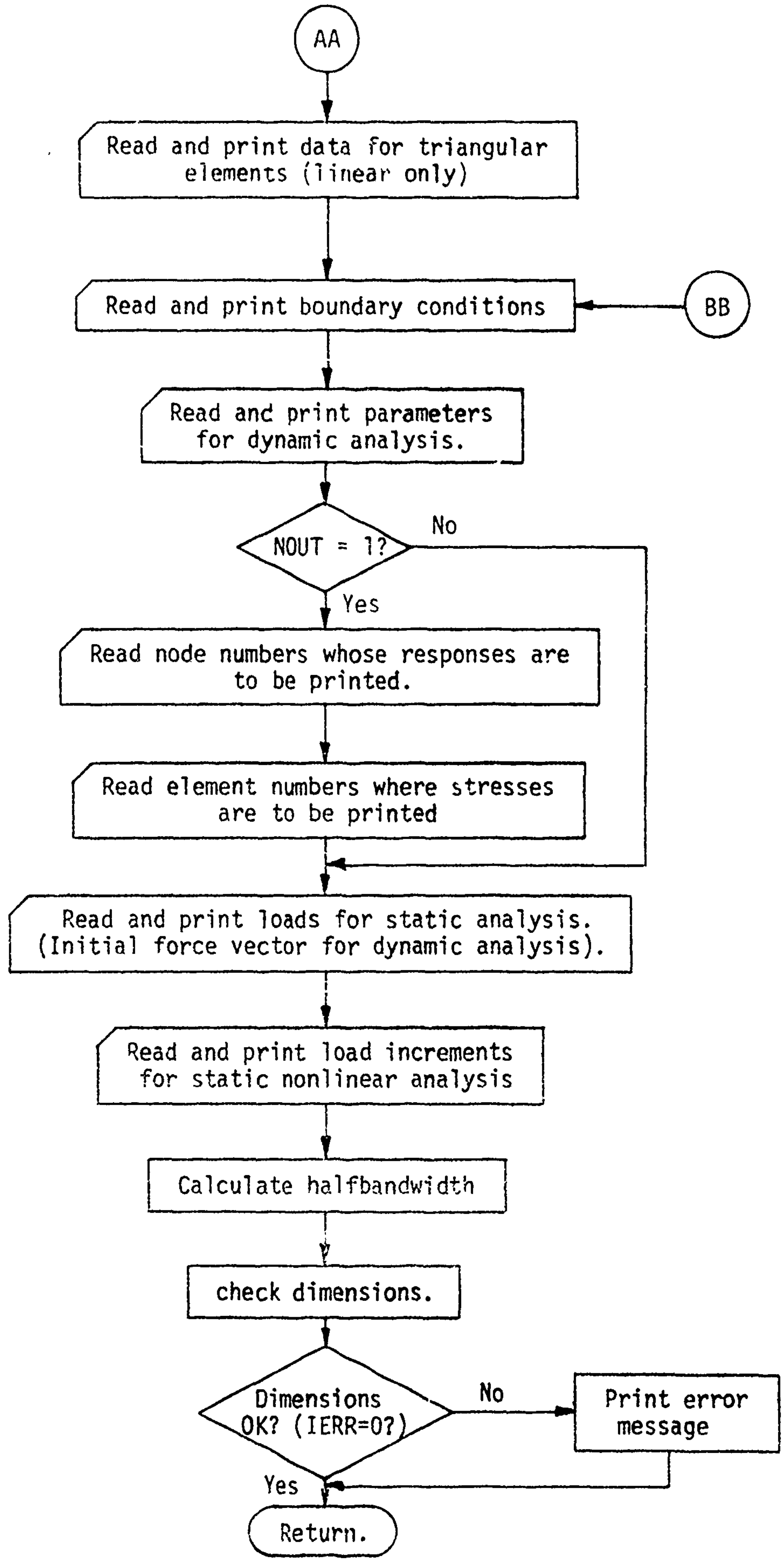

AA
Dynamic problem neglecting dead loads?
Yes
CC
No
Read AK from Tape 1
Call BCOND (to incorporate boundary conditions into AK)
Call DECOMP (to decompose AK for equation solving)
Select full static load, FF, (with or w/o dead loads) and print headings for static solution
Call STSOL (to obtain and print linear solution at full static load)
Call RESET (to check for nonlinear response and indicate with KCHAK if an incremental static solution is needed)
KCHAK = 0?
Yes
No
NT = NT + 1
Call STNON (to perform incremental static solution)
BB


BB
Static Problem only?
Yes
DD
No
NS = NS + 1
CC
Print NS with heading for next time step
First dynamic Step (NS=1)?
Yes
No
KCHAK = 0?
Yes
No
Form left-hand-side matrix, AK for dynamic analysis
Call BCOND (to incorporate boundary conditions into AK)
Call DECOMP (to decompose AK)
Write decomposed AK on Tape 2
Read or calculate forces for this time step, FF, and print (Call FORGEN if FF calculated)
Call SOLVE (to obtain and print dynamic response at this step)
EE

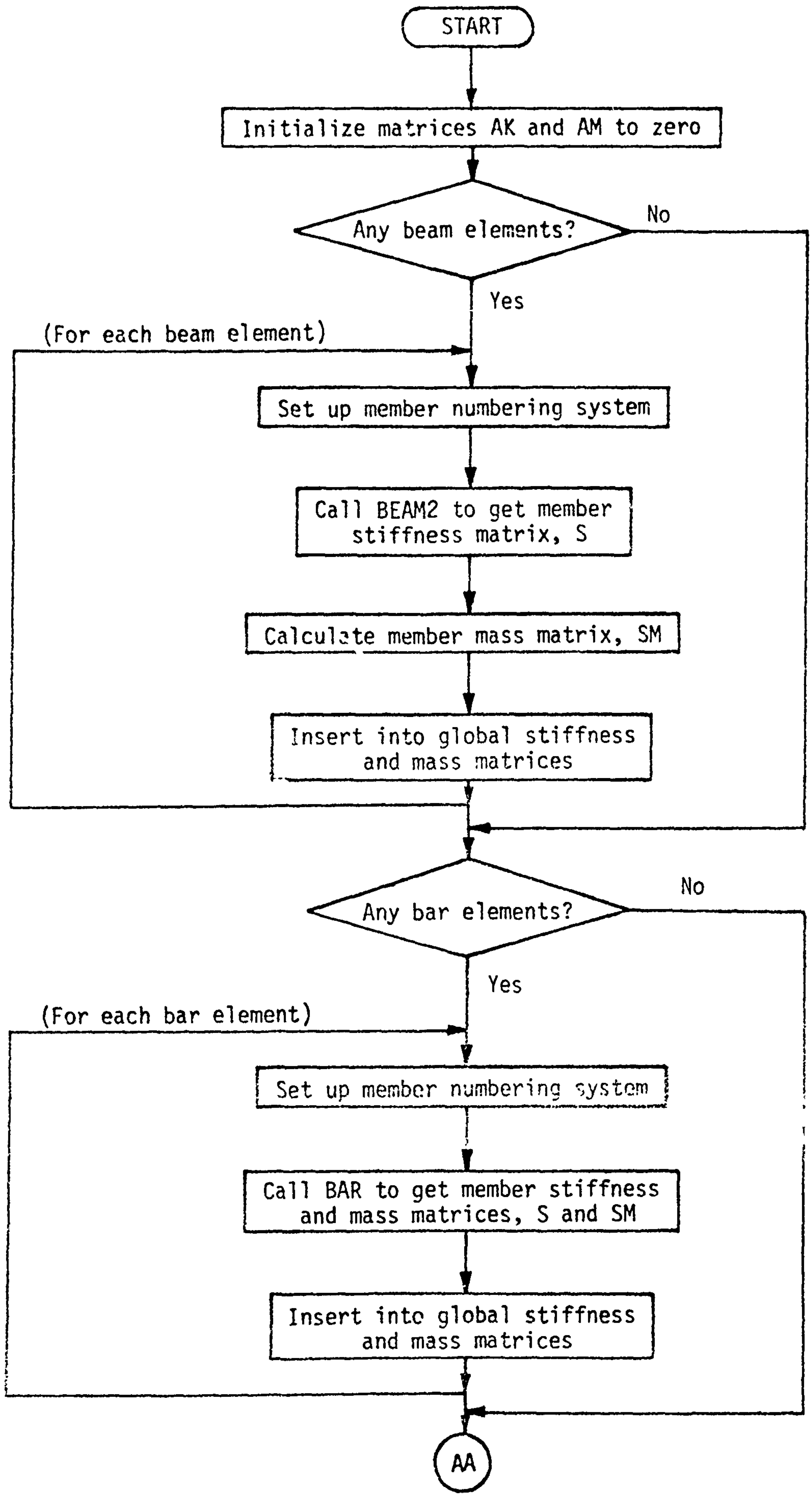

EE
Linear Problem and no member stresses desired?
Yes
No
Call RESET (to check for nonlinear response and modify matrix AK accordingly)
Yes
No
DD
NS > NSMAX?
CC
(return for next problem)
(Continue with next time step)

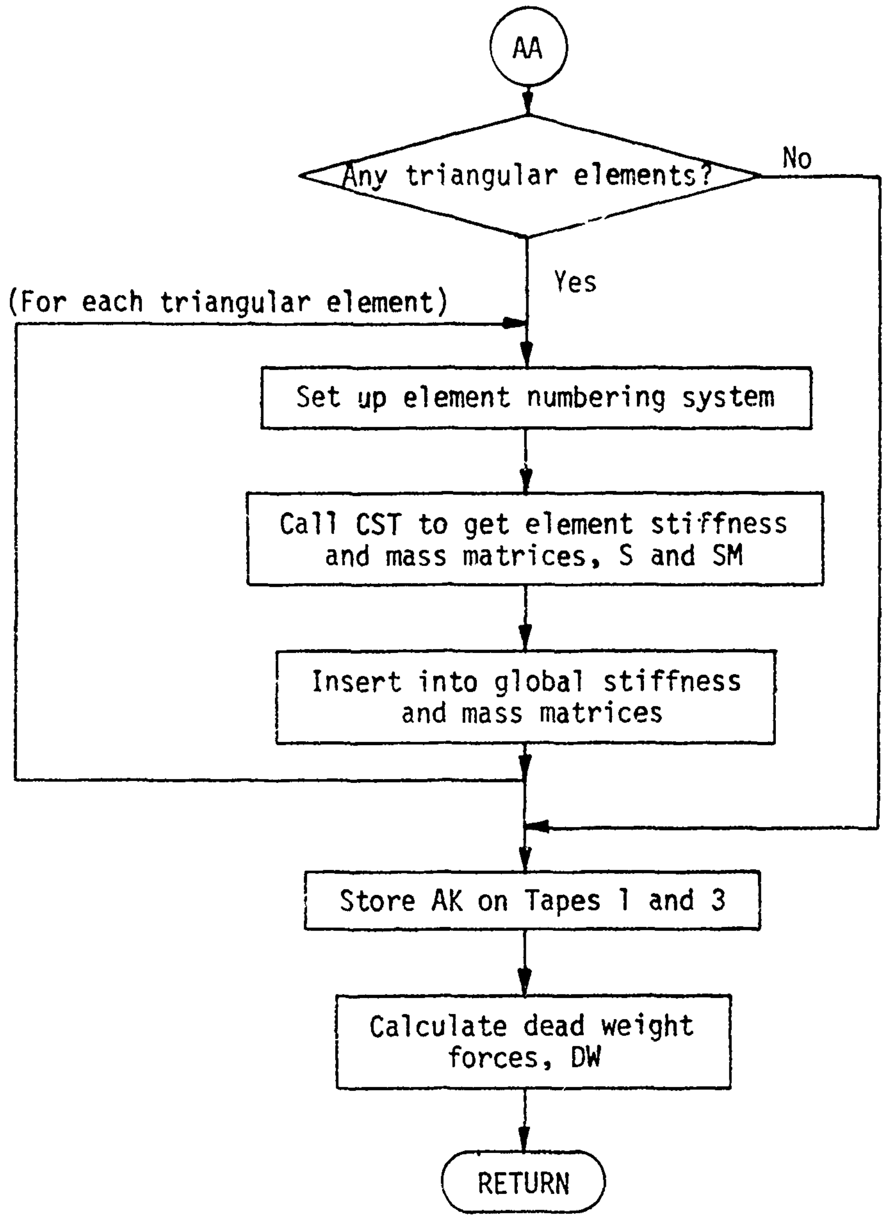
B. Subroutine SETUP



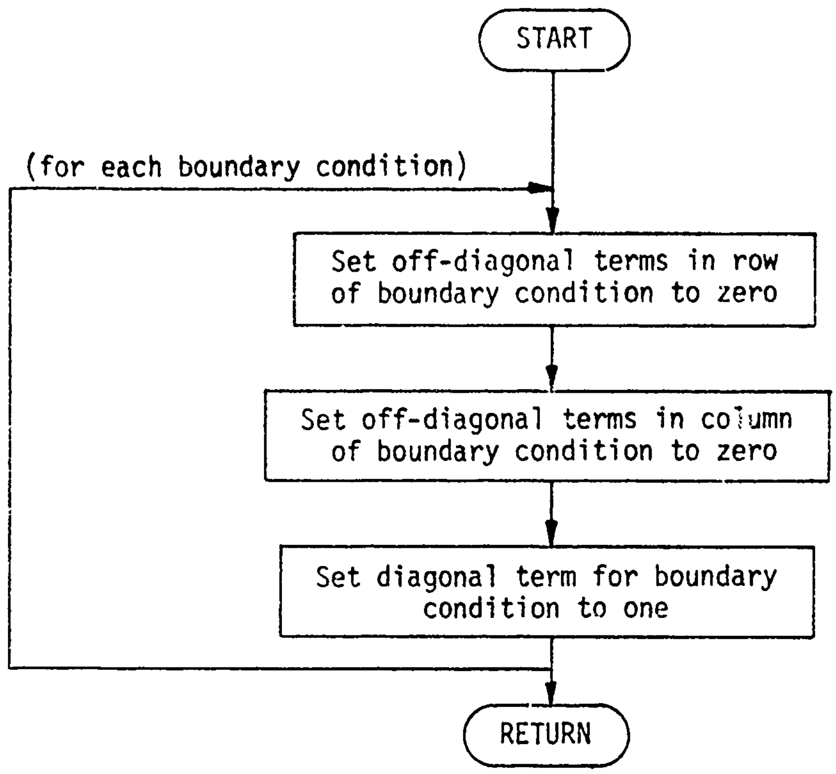

AA
Read and print data for triangular elements (linear only)
Read and print boundary conditions
BB
Read and print parameters for dynamic analysis.
NOUT = 1?
No
Yes
Read node numbers whose responses are to be printed.
Read element numbers where stresses are to be printed
Read and print loads for static analysis. (Initial force vector for dynamic analysis).
Read and print load increments for static nonlinear analysis
Calculate halfbandwidth
check dimensions.
Dimensions OK? (IERR=0?)
No
Print error message
Yes
Return.

C. SUBROUTINE ASMBLE

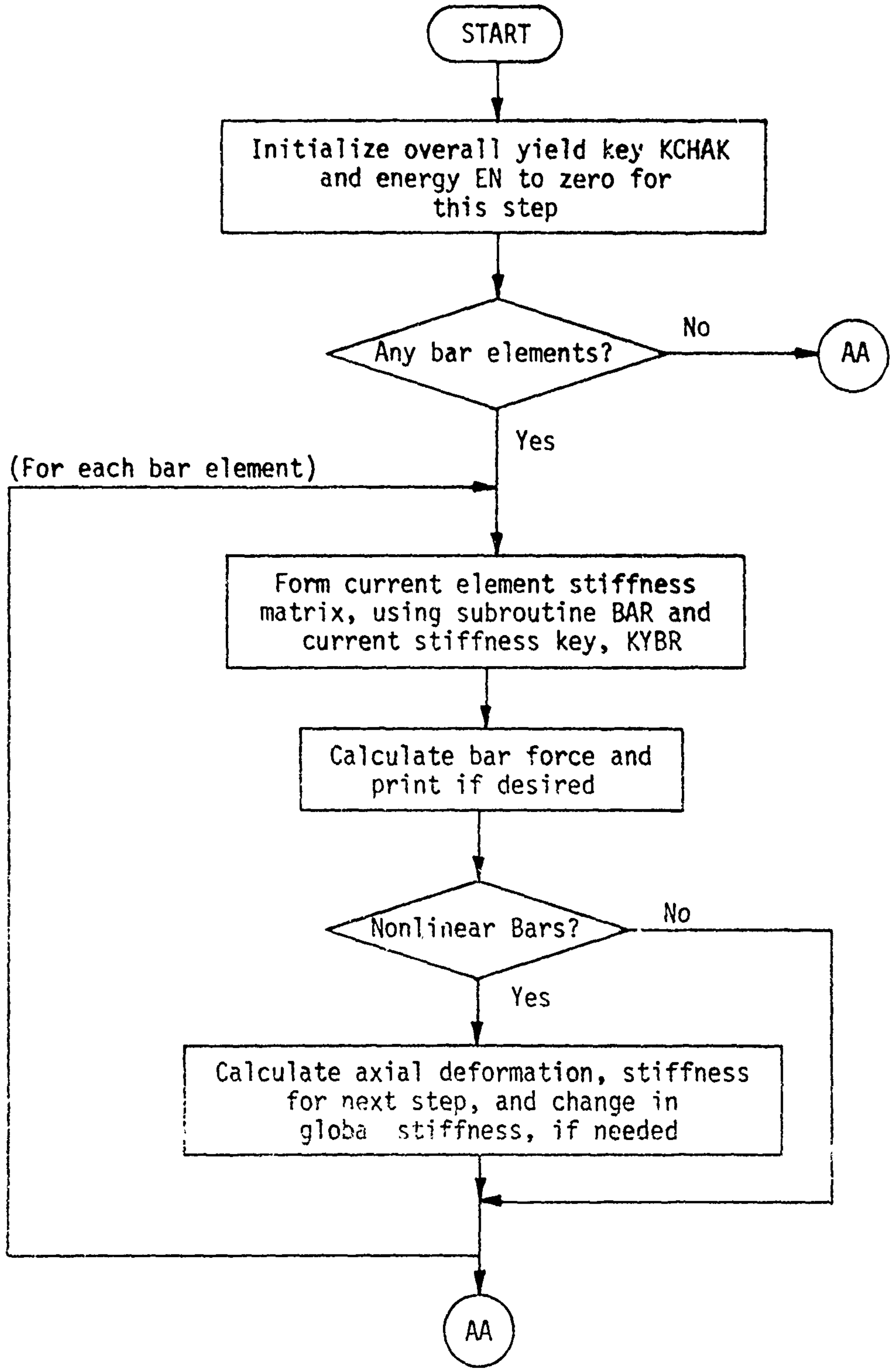


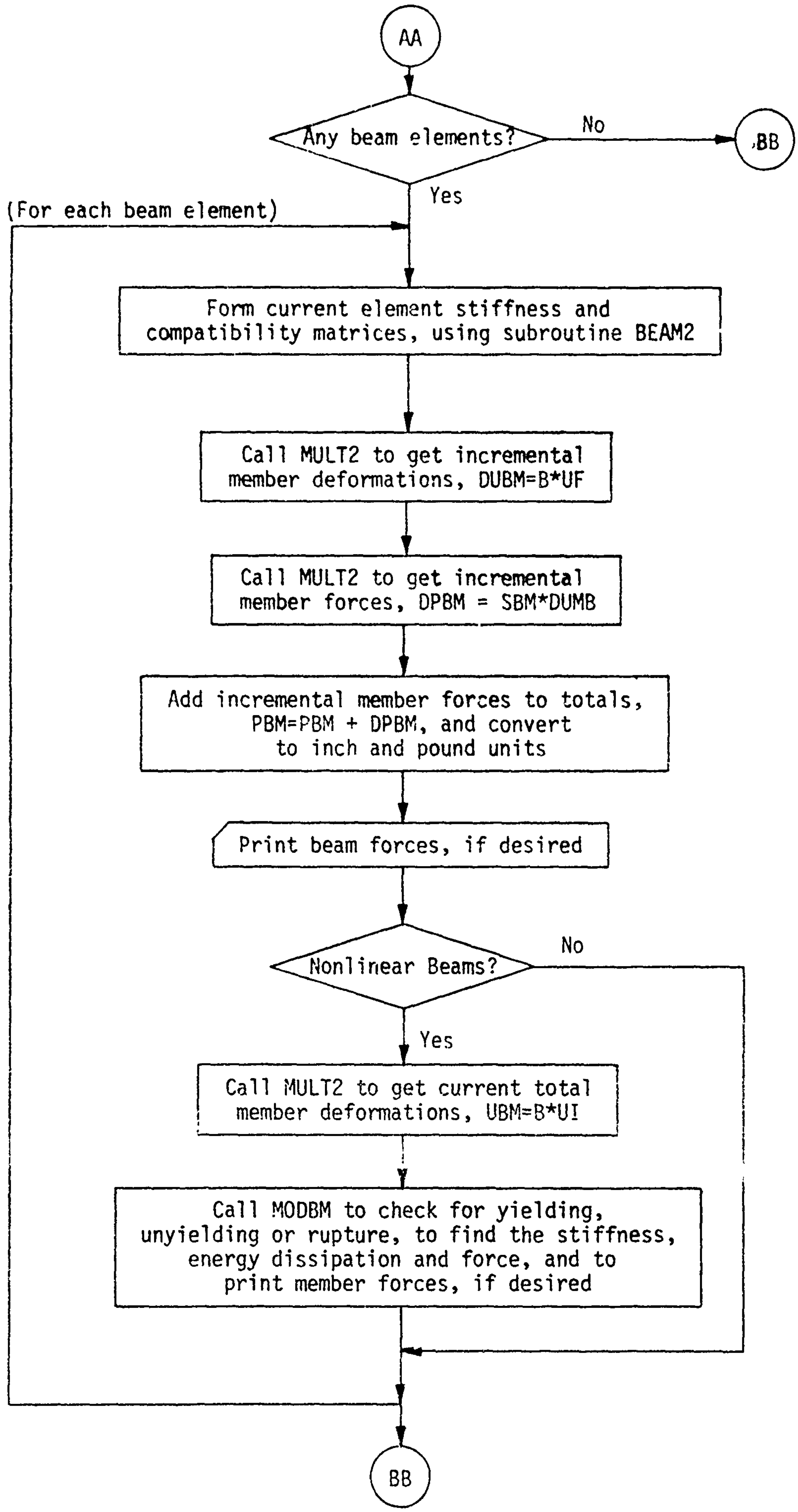

AA
Any triangular elements?
No
Yes
(For each triangular element)
Set up element numbering system
Call CST to get element stiffness and mass matrices, S and SM
Insert into global stiffness and mass matrices
Store AK on Tapes 1 and 3
Calculate dead weight forces, DW
RETURN

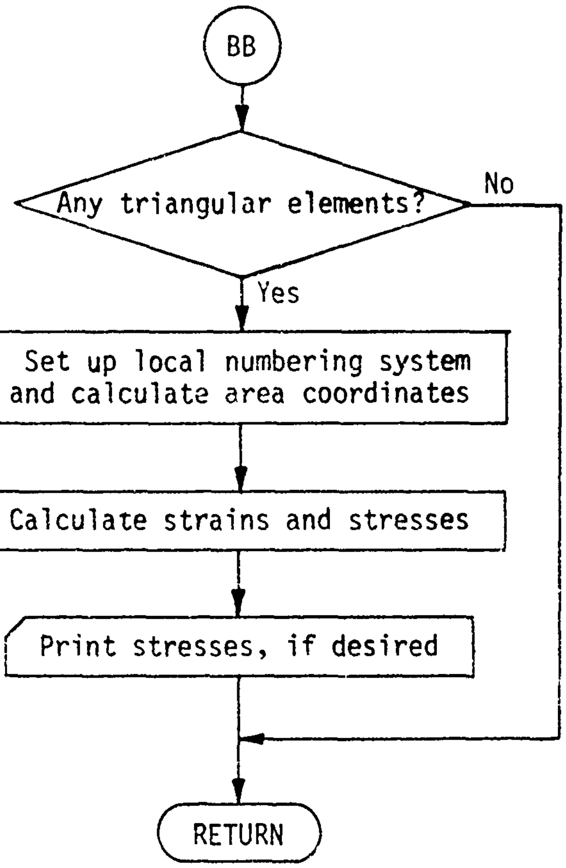
D. Subroutine BCOND



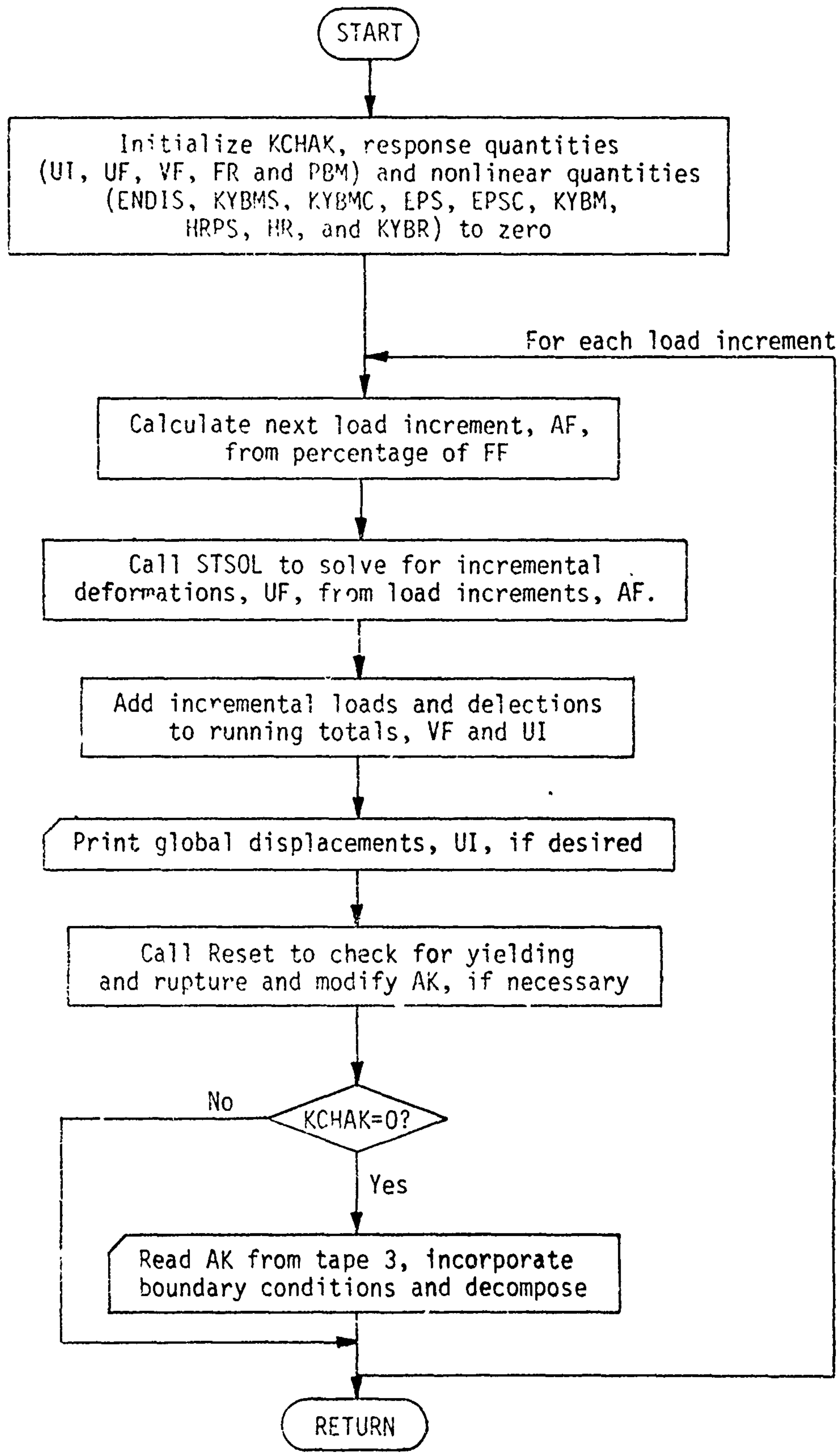
G. Subroutine RESET



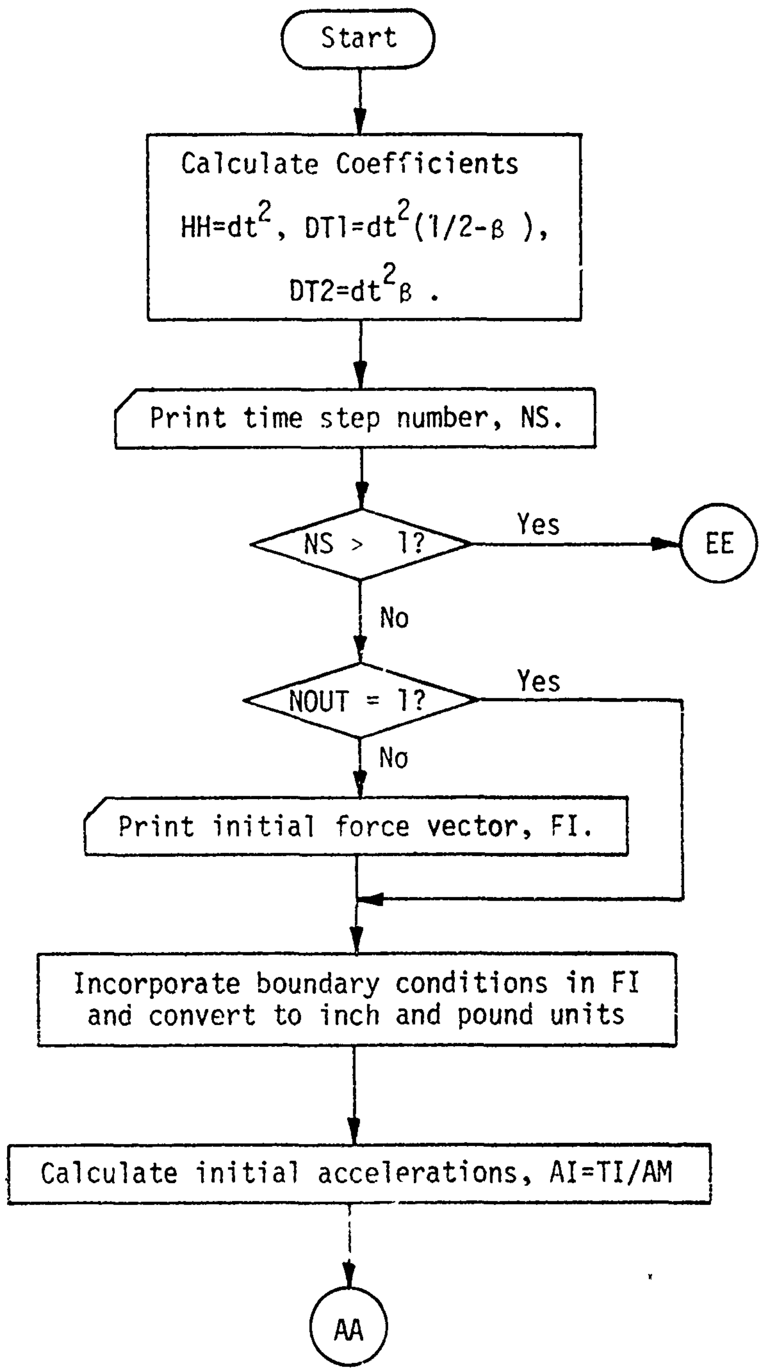

AA
Any beam elements?
No
BB
Yes
(For each beam element)
Form current element stiffness and compatibility matrices, using subroutine BEAM2
Call MULT2 to get incremental member deformations, DUBM=B*UF
Call MULT2 to get incremental member forces, DPBM = SBM*DUMB
Add incremental member forces to totals, PBM=PBM + DPBM, and convert to inch and pound units
Print beam forces, if desired
Nonlinear Beams?
No
Yes
Call MULT2 to get current total member deformations, UBM=B*UI
Call MODBM to check for yielding, unyielding or rupture, to find the stiffness, energy dissipation and force, and to print member forces, if desired
BB

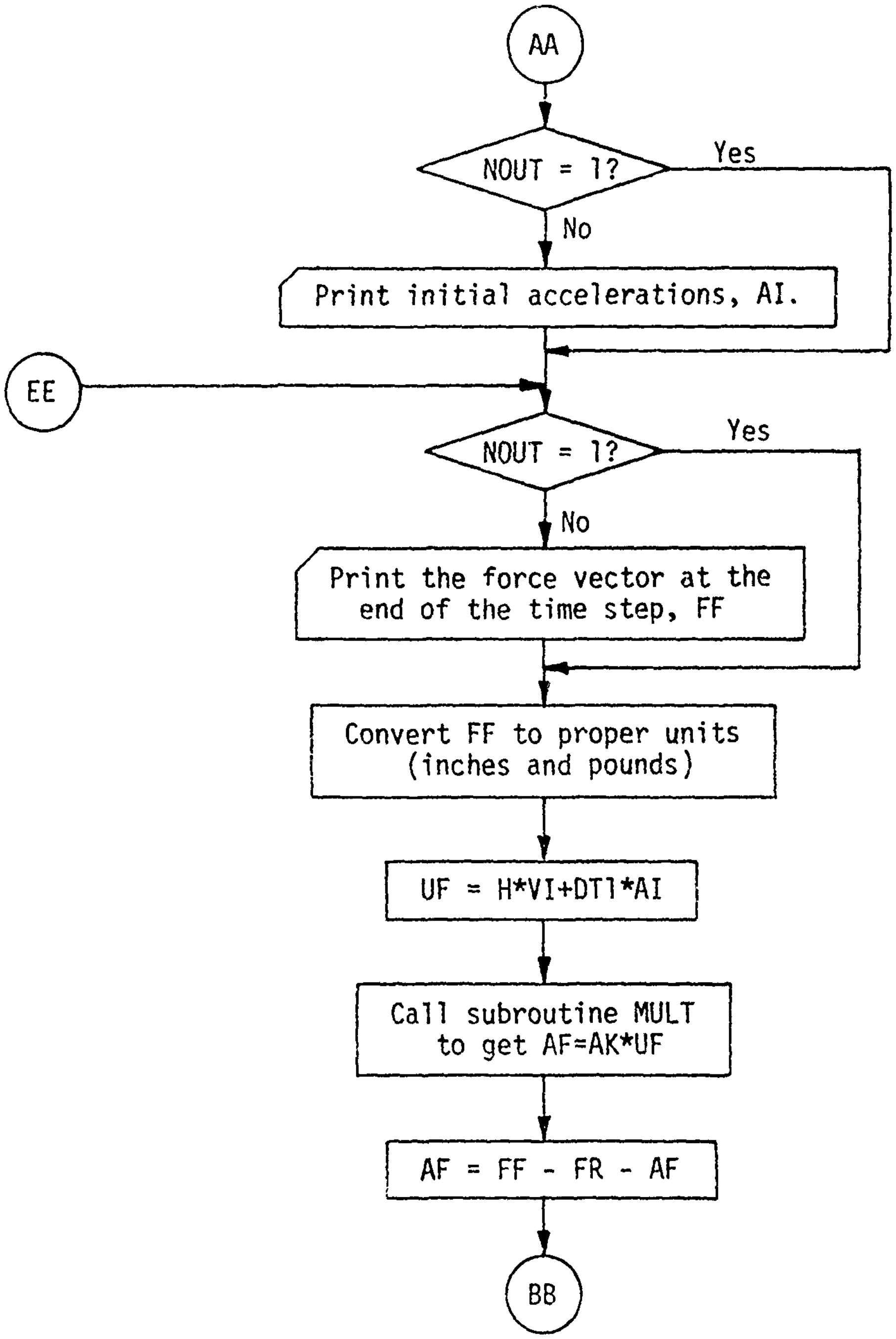

BB
Any triangular elements?
No
Yes
Set up local numbering system and calculate area coordinates
Calculate strains and stresses
Print stresses, if desired
RETURN

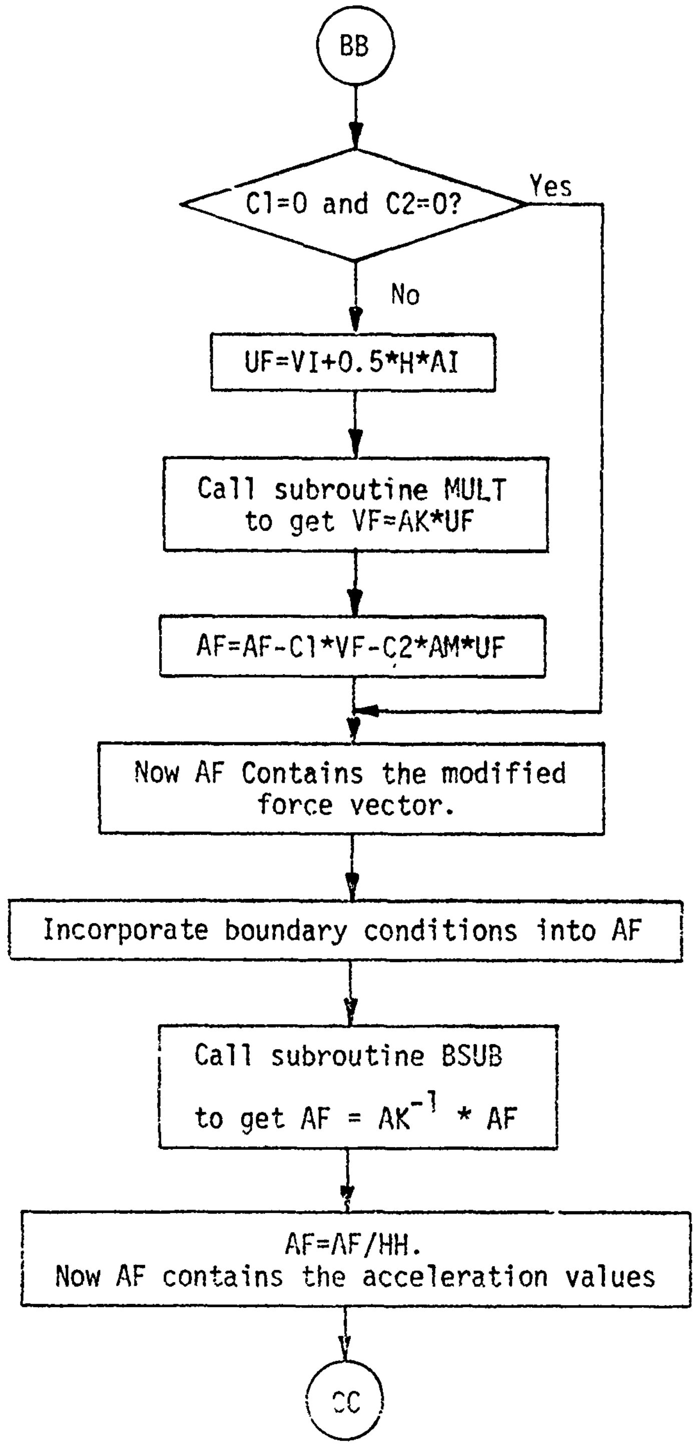
H. Subroutine STNON

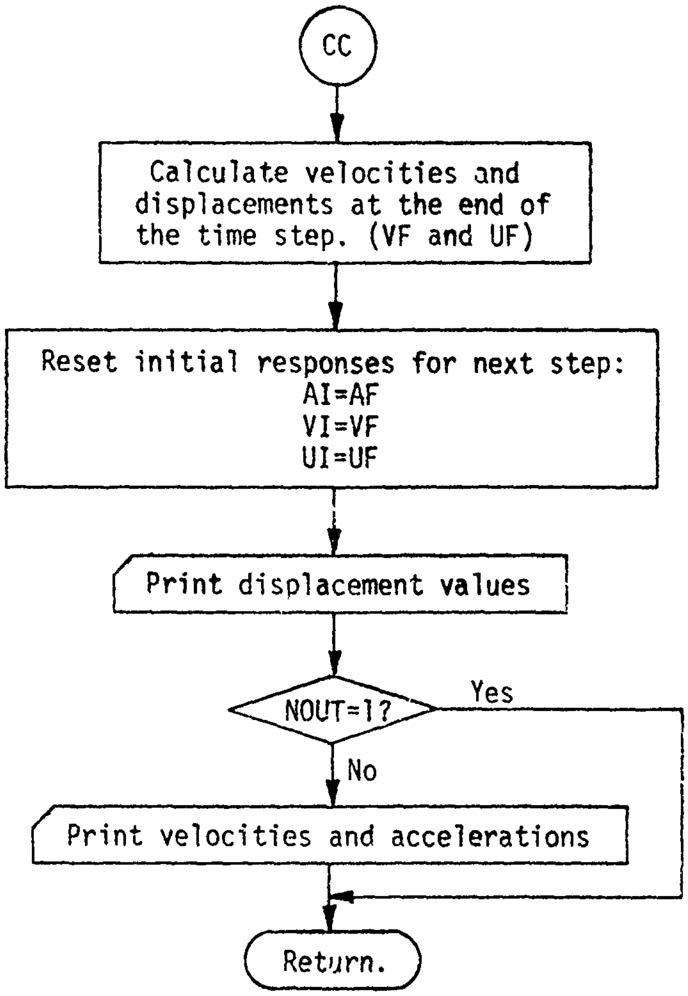


J. Subroutine SOLVE.

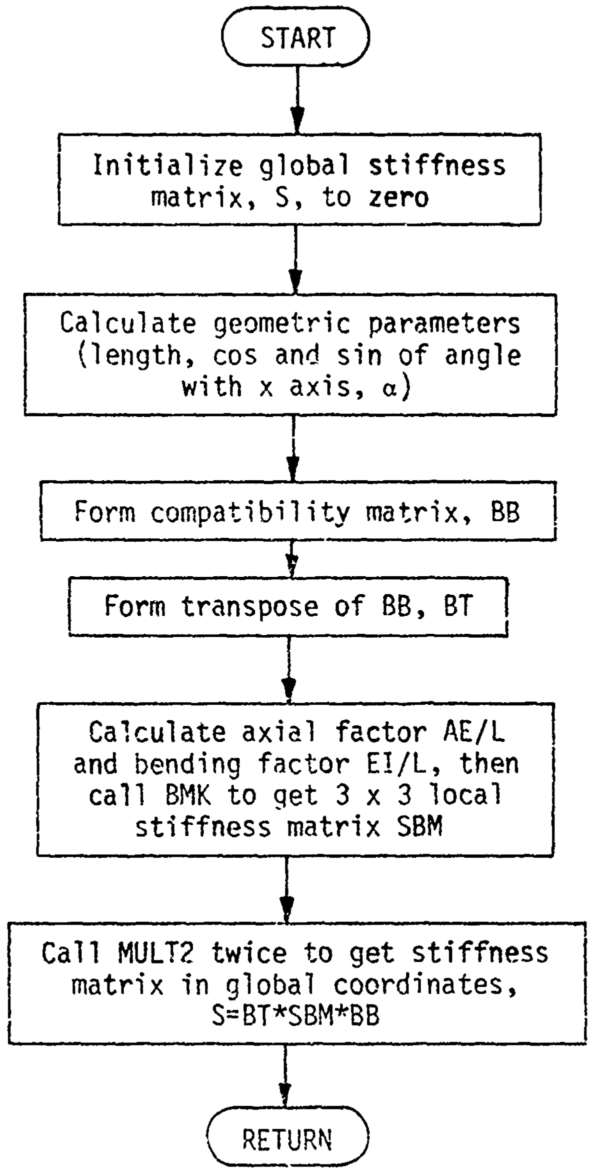



AA
NOUT = 1?
Yes
No
Print initial accelerations, AI.
EE
NOUT = 1?
Yes
No
Print the force vector at the end of the time step, FF
Convert FF to proper units (inches and pounds)
UF = H*VI+DT1*AI
Call subroutine MULT to get AF=AK*UF
AF = FF - FR - AF
BB

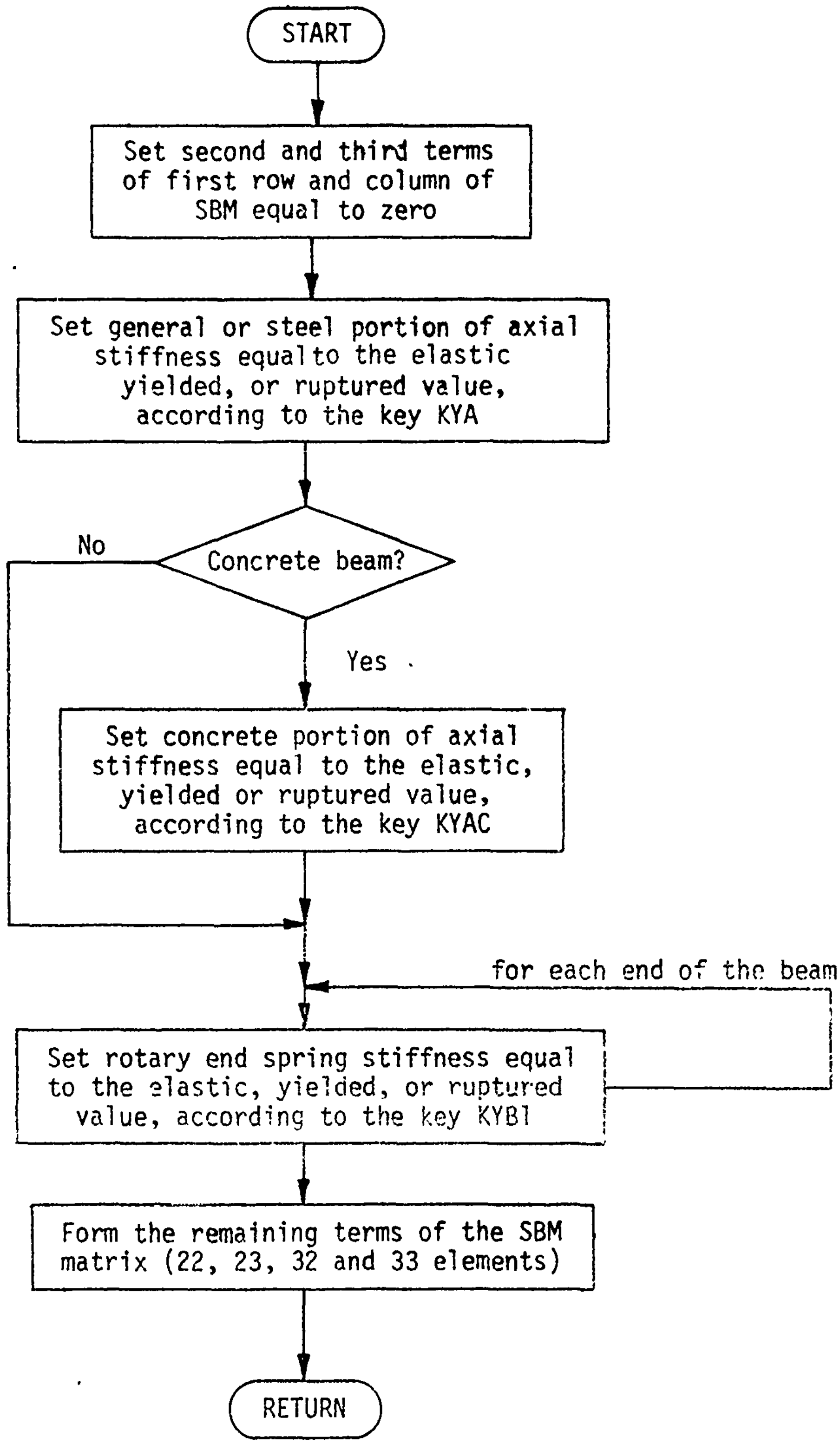

BB
C1=0 and C2=0?
Yes
No
UF=VI+0.5*H*AI
Call subroutine MULT to get VF=AK*UF
AF=AF-C1*VF-C2*AM*UF
Now AF Contains the modified force vector.
Incorporate boundary conditions into AF
Call subroutine BSUB to get AF = AK^{-1} * AF
AF=AF/HH.
Now AF contains the acceleration values
CC


CC
Calculate velocities and displacements at the end of the time step. (VF and UF)
Reset initial responses for next step:
AI=AF
VI=VF
UI=UF
Print displacement values
NOUT=1?
Yes
No
Print velocities and accelerations
Return.

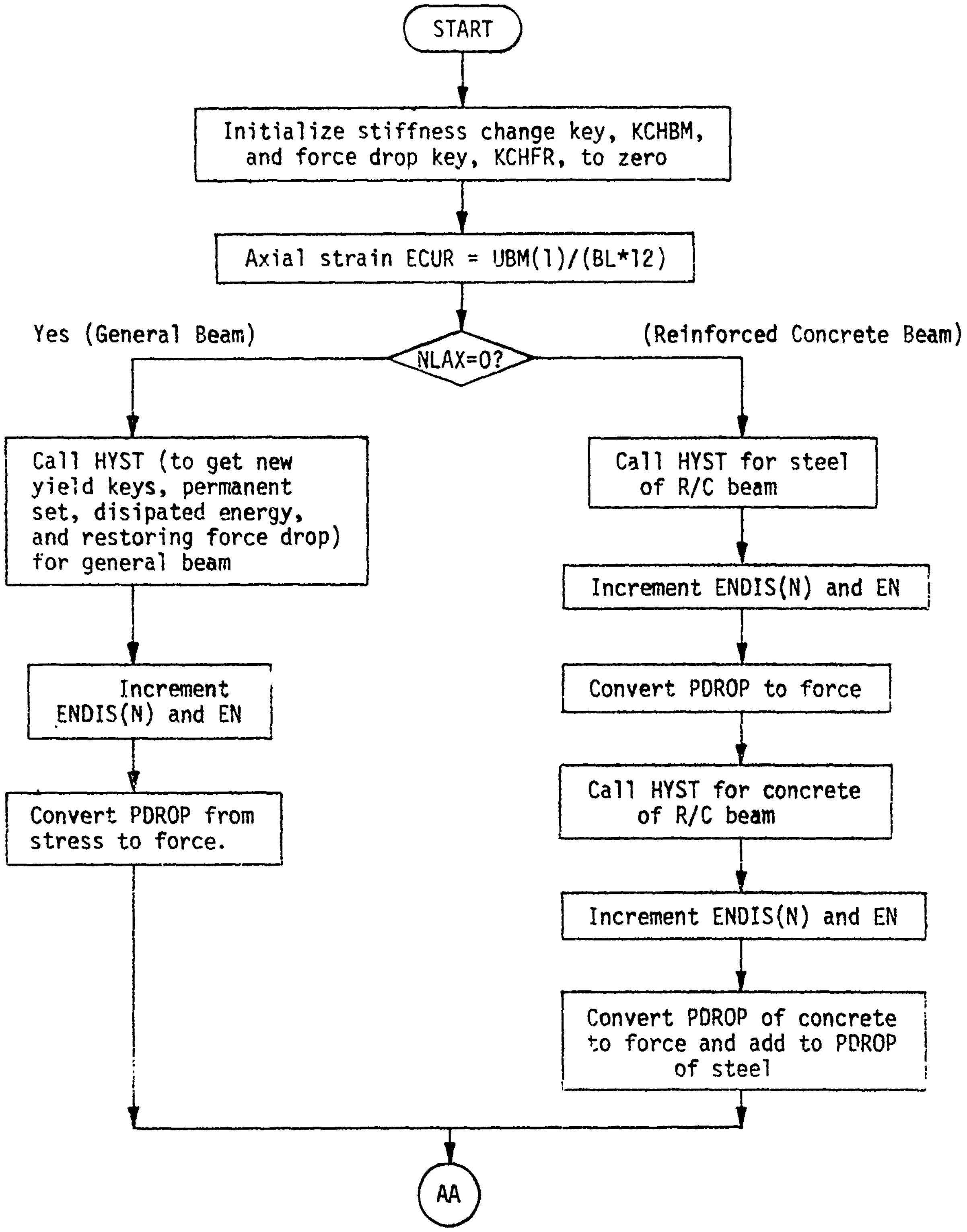
K. Subroutine BEAM2

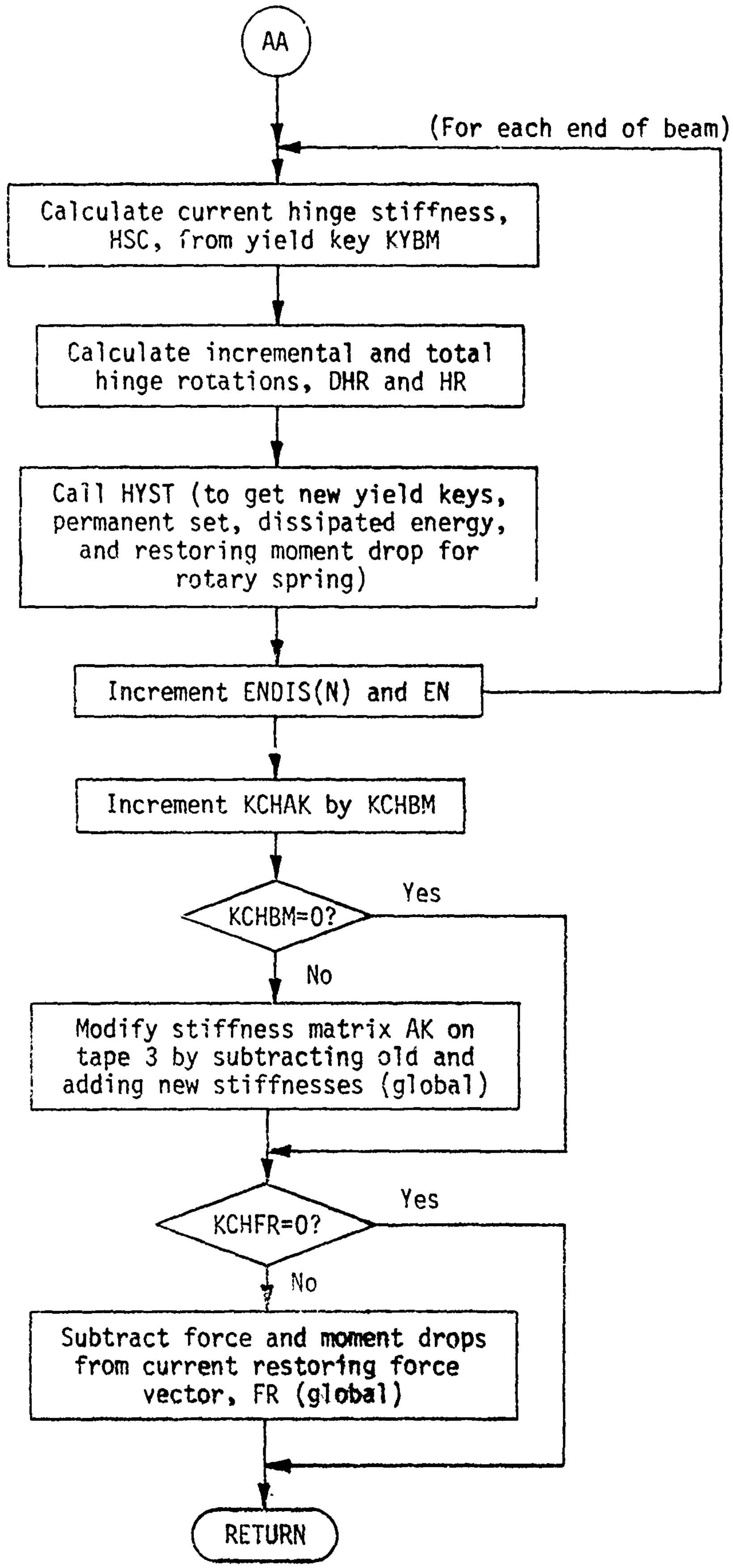


L. Subroutine BMK

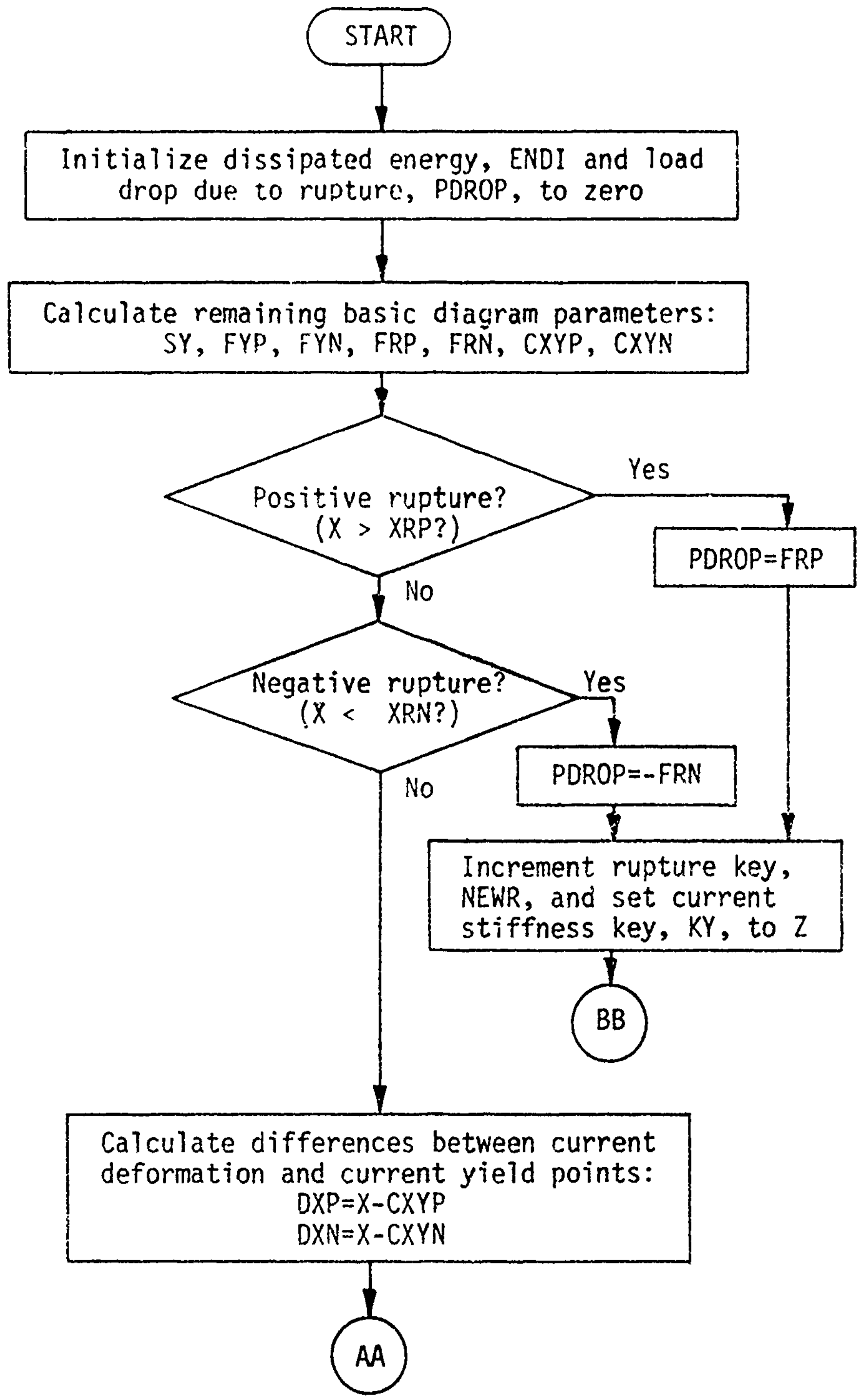


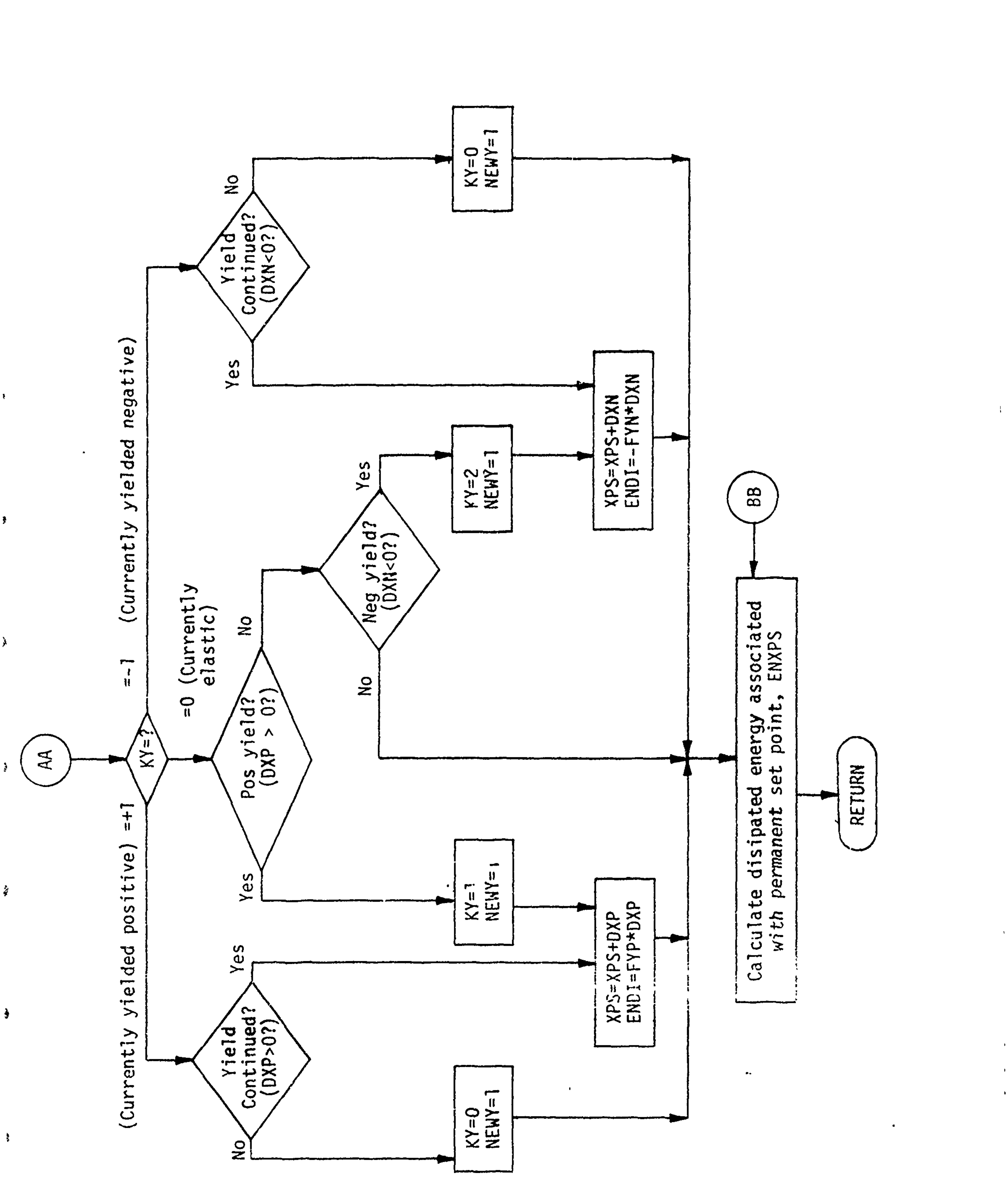
P. Subroutine MODBM




AA
(For each end of beam)
Calculate current hinge stiffness, HSC, from yield key KYBM
Calculate incremental and total hinge rotations, DHR and HR
Call HYST (to get new yield keys, permanent set, dissipated energy, and restoring moment drop for rotary spring)
Increment ENDIS(N) and EN
Increment KCHAK by KCHBM
KCHBM=0?
Yes
No
Modify stiffness matrix AK on tape 3 by subtracting old and adding new stiffnesses (global)
KCHFR=0?
Yes
No
Subtract force and moment drops from current restoring force vector, FR (global)
RETURN

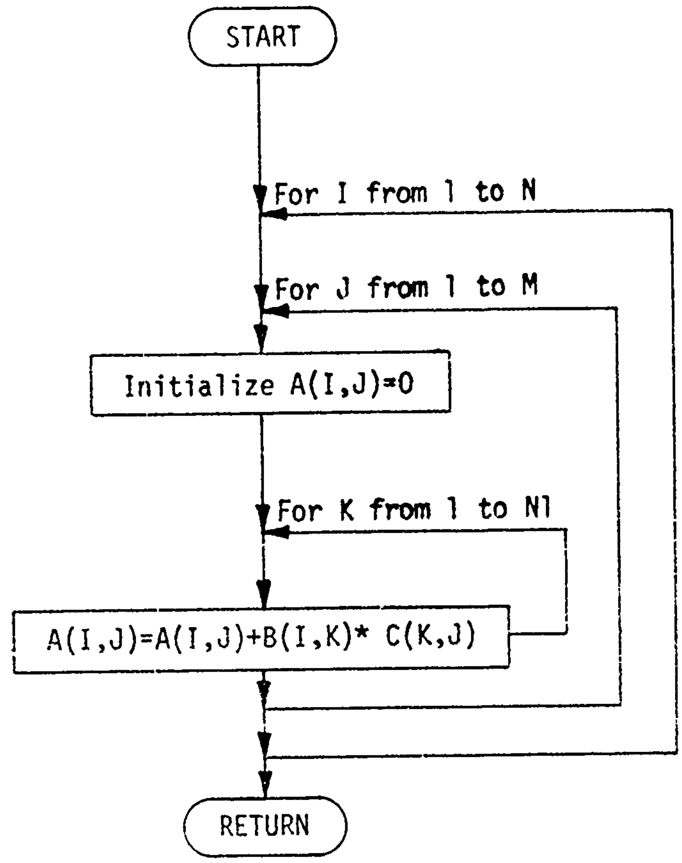
Q. Subroutine HYST



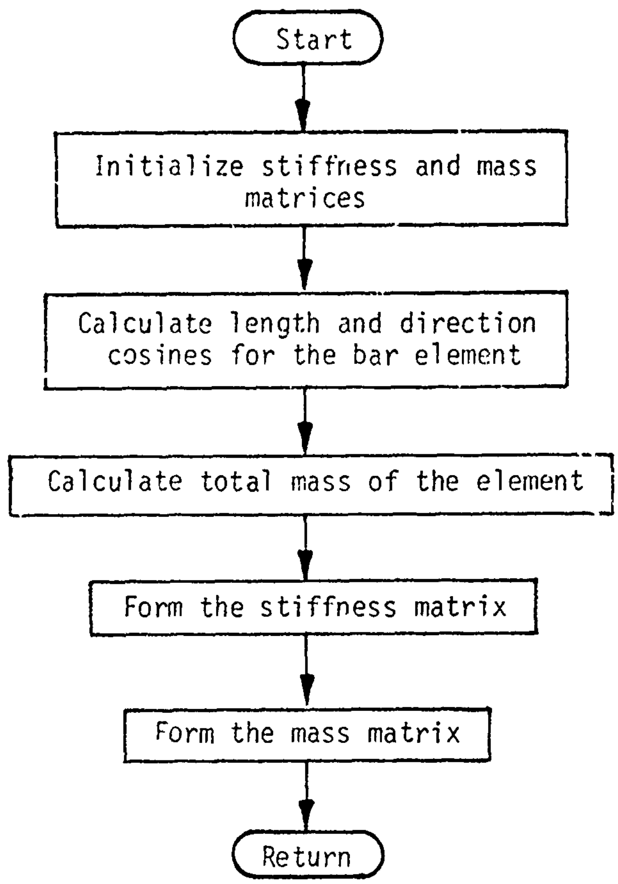


AA
KY=?
(Currently yielded positive) =+1
=-1 (Currently yielded negative)
=0 (Currently elastic)
Yield Continued? (DXP>0?)
Yes
No
KY=0 NEWY=1
XPS=XPS+DXP ENDI=FYP*DXP
Pos yield? (DXP > 0?)
Yes
No
KY=1 NEWY=1
Neg yield? (DXN<0?)
Yes
No
KY=2 NEWY=1
XPS=XPS+DXN ENDI=-FYN*DXN
Yield Continued? (DXN<0?)
Yes
No
KY=0 NEWY=1
BB
Calculate dispated energy associated with permanent set point, ENXPS
RETURN

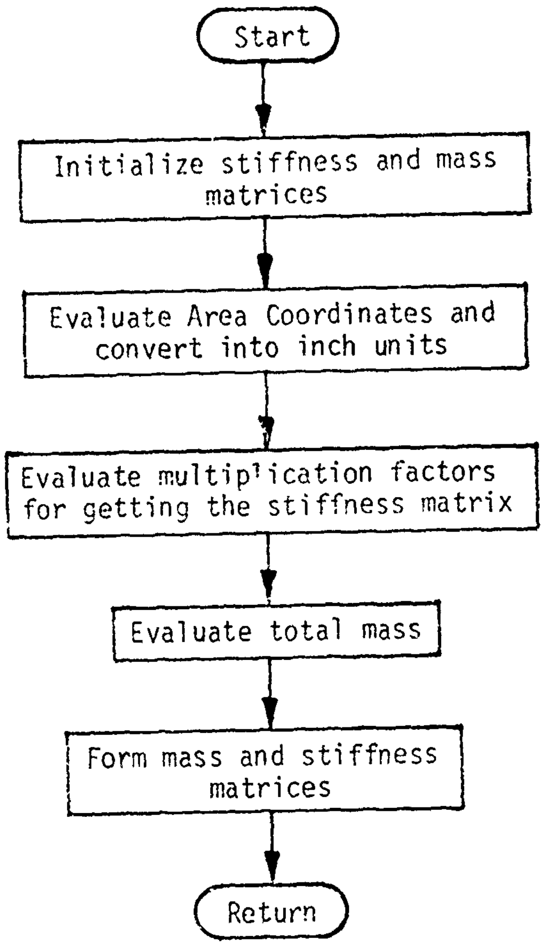
S. Subroutine MULT2



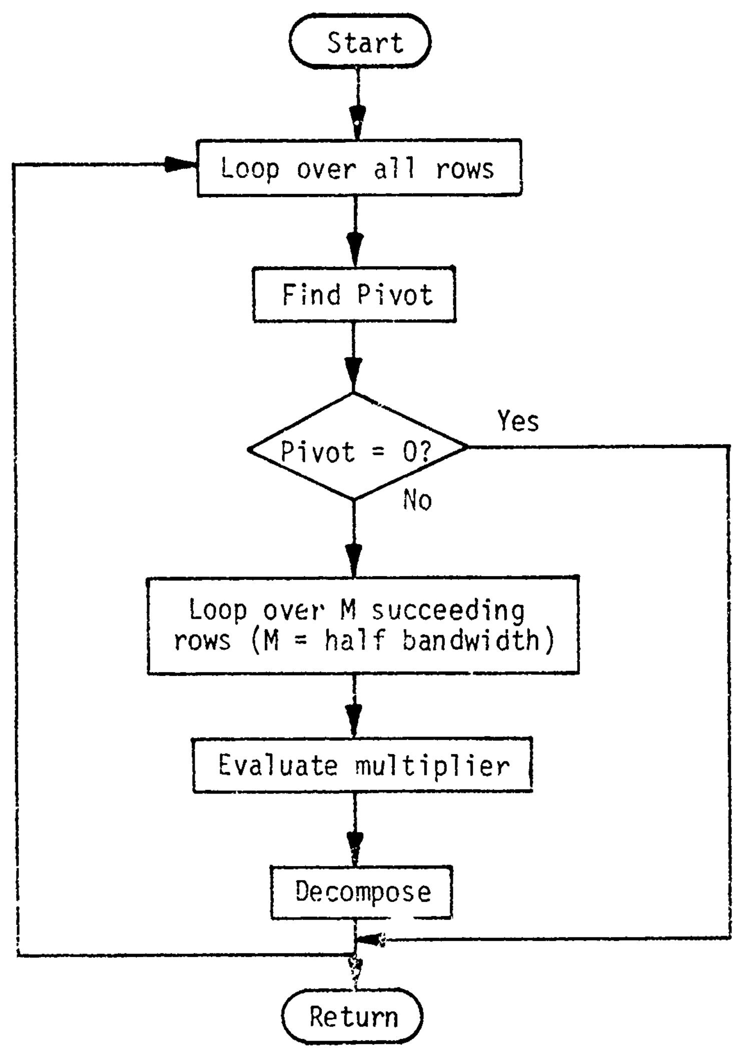
H. Subroutine BAR.



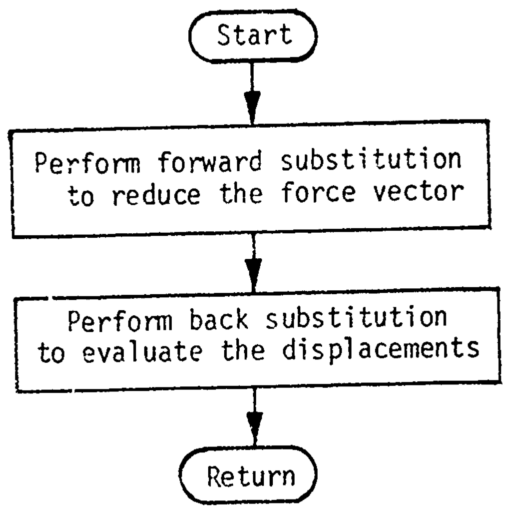
I. Subroutine CST.



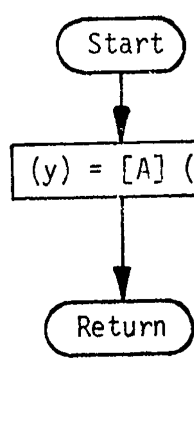
J. Subroutine DECOMP



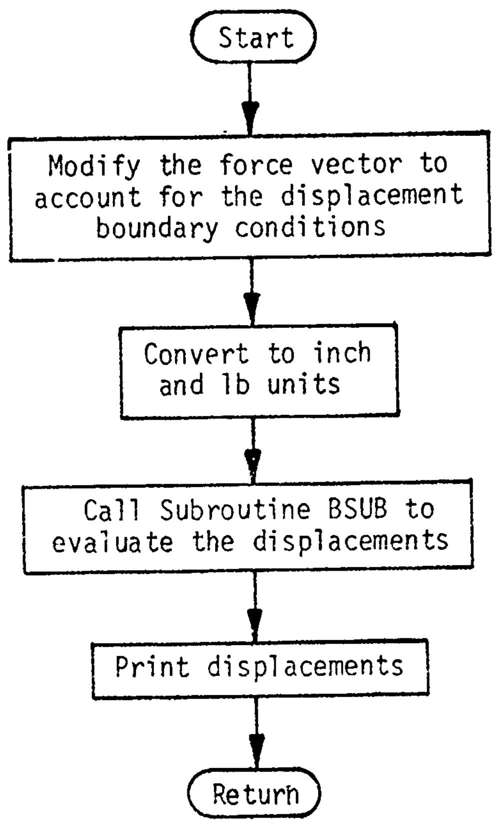
K. Subroutine BSUB.



L. Subroutine MULT.



M. Subroutine STSOL.



V. DESCRIPTION OF THE MAJOR VARIABLE NAMES

Many of the major variables are grouped into eight COMMON statements. The same variable names are used in the MAIN program and the major subroutines called by MAIN, namely, subroutines SETUP, ASMBLE, STSOL, RESET, STNON, and SOLVE.

A. Variables in the First COMMON Block (Integer, Dimensions and Options)

Variable Name	Description
NNOD	Number of nodes
NBMEL	Number of beam elements
NBREL	Number of bar elements
NTREL	Number of triangular elements
NROW	Number of rows of the stiffness matrix and mass matrix
NHB	Half band width (including diagonal of the stiffness matrix
NBOU	Number of nodes with prescribed displacement boundary conditions
IBOU	Total number of prescribed boundary conditions
NSMAX	Maximum number of time steps for dynamic analysis
NMAX	Available computer storage storage for the stiffness matrix. (The value of NMAX depends upon the computer and the dimensions of other variables. This value should be specified in the MAIN program.
IERR	Error flag

IERR	NATURE OF ERROR
101	Number of beam elements exceeds dimensions provided
102	Number of bar elements exceeds dimensions provided
103	Number of triangular elements exceeds dimensions provided

104	Number of rows of the matrices exceeds dimensions provided
105	Matrix size is greater than NMAX (available core storage)
106	Number of boundary conditions exceeds dimensions provided
NTYPE	Option parameter to specify static or dynamic analysis
NLIN	Option parameter specifying linear or nonlinear analysis
NRES	Option parameter specifying whether or not stresses are to be determined at each step
NOUT	Option parameter controlling the output
NNDO	Total number of special nodes where responses are chosen to be printed out
NNBL	Total number of special beam elements whose forces are to be printed out
NNBR	Total number of special bar elements whose forces are to be printed out
NNTR	Total number of special triangular elements whose stresses are to be printed out
NT	Counter for nonlinear static load increments

B. Variables in the Second COMMON Block (Dynamic Parameters)

Variable Name	Description
C1	Constant to multiply the stiffness matrix to get one part of the damping matrix
C2	Constant to multiply the mass matrix to get the other part of the damping matrix
H	Magnitude of the time increment for dynamic analysis
BETA	Newmark's β parameter

C. Variables in the Third COMMON Block (Element Node Numbers)

Variable Name	Description
NDBM	Beam element node numbers
NDBR	Bar element node numbers

NDTR	Triangular element node numbers
NDO	Special nodes for output
NBL	Special beam elements for print-out
NBR	Special bar elements for print-out
NTR	Special triangular elements for print-out
NBO	Specified boundary conditions

D. Variables in the Fourth COMMON Block (Vectors of Linear System Properties

Variable Name	Description
X	x coordinates of nodes, positive to right
Y	y coordinates of nodes, positive up
EBM	Moduli of elasticity of beam elements
ABM	Cross-sectional areas of beam elements
BMI	Moments of inertia of beam elements
BMW	Unit weights of beam elements
EBR	Moduli of elasticity of bar elements
ABR	Cross-sectional areas of bar elements
BRW	Unit weights of bar elements
ETR	Moduli of elasticity for triangular elements
PRTR	Poisson's ratios for triangular elements
THTR	Thicknesses of triangular elements
TRW	Unit weights of triangular elements
AK	Stiffness matrix
AM	Mass matrix
UI	Displacements at the beginning of a time step (also displacements in static analysis)
VI	Velocities at the beginning of a time step
AI	Accelerations at the beginning of a time step
FI	Forces at the beginning of a time step (also forces for static analysis)
UF	Displacements at the end of a time step
VF	Velocities at the end of a time step
AF	Accelerations at the end of a time step

FF	Forces at the end of a time step
DW	Dead weight forces
S	Stiffness matrix of an element. (Same variable name is used for the beam, bar and triangular elements)
SM	Mass matrix of an element (Same name for all elements)
PBM	Forces and moments in beam elements
PBR	Axial forces in bar elements
PTR	Stresses in triangular elements
PBBM	Beam element forces and moments due to dead load
PBBR	Bar element forces due to dead load
PBTR	Triangle element stresses due to dead load

E. Variables in the Fifth COMMON Block (Nonlinear Options and Parameters)

Variable Name	Description
NLEXT	Option parameter to specify if beams, bars or both are nonlinear
NLAX	Option parameter to specify homogeneous or reinforced concrete beams
ISPRT	Option parameter allowing reading or interval prescription of static load increments
ES	Modulus of elasticity of steel
CRACK	Percentage of compressive yield load at tensile cracking of concrete
NPCTLD	Number of incremental static load percentages specified
BL	Beam or bar length
NDEAD	Option parameter specifying whether dead loads are included or neglected
EN	Total dissipated energy in a structure

F. Variables in the Sixth COMMON Block (Nonlinear Element Properties and Keys-Vectors and Matrices)

Variable Name	Description
EYS	Yield strain of steel in a R/C beam
ERS	Rupture strain of steel in a R/C beam
SSBMS	Second slope ratio of beam steel
AS	Area of beam steel
EYC	Yield strain of concrete in a R/C beam
ERC	Rupture strain of concrete in a R/C beam
SSBMC	Second slope ratio of beam concrete
HS	Rotary hinge stiffness
HRYP	Positive yield value of hinge rotation
HRYN	Negative yield value of hinge rotation
HRRP	Positive rupture value of hinge rotation
HRRN	Negative rupture value of hinge rotation
SSBM	Second slope ratio of beam hinge
EPS	Strain at permanent set point of steel
KYBMS	Yield key for beam steel
EPSC	Strain at permanent set point of concrete
KYBMC	Yield key for beam concrete
HRPS	Hinge rotation at permanent set point
KYBM	Yield key for beam hinge
ENDIS	Dissipated energy in a beam
SSBR	Second slope ratio for a bar
FR	Restoring force
HR	Hinge rotation
KYBR	Yield key for a bar

G. Variables in the Seventh COMMON Block

Variable Name	Description
PCTLD	Percentage of full static load in an incremental static solution
SBM	Local stiffness matrix of a beam element (3 x 3)
BB	Compatibility matrix of a beam element (3 x 6)
BT	Transpose of BB
VBM	Local deformations of a beam element
VBR	Local deformation of a bar element

H. Variables not included in COMMON statements:

Additional variables used in the MAIN program and the subroutines are described below.

1. MAIN

Variable Name	Description
TITLE	Problem title
NPROB	Problem number
NS	Time step number for dynamic analysis

2. Subroutine SETUP

Variable Name	Description
Form	Variable used for object-time-format for print-out of prescribed boundary conditions
IBC, BC	Work areas used for reading in boundary conditions and loads.

3. Subroutine ASMBLE

Variable Name	Description
KK, XX, YY	Work areas used for assembling the stiffness and mass matrices

4. Subroutine BCOND

Variable Name	Description
A	Matrix to have boundary conditions incorporated (stored as a singly subscripted variable)
NROW	Number of rows of A
M	Half band with (including diagonal) of A
NA	NROW * M = Total storage for A

5. Subroutine DECOMP

Variable Name	Description
A	Matrix to be decomposed
NROW, M, NA	As in Subroutine BCOND

6. Subroutine STSOL

Variable Name	Description
A, NROW, M, NA	Same as in subroutines BCOND and DECOMP
X	Solution vector (Force vector on entering the subroutine, Displacement vector when coming out of the subroutine).
F	Force vector corresponding to the static loads

7. Subroutine RESET

Variable Name	Description
SC	Scale factor used for converting from inches and pounds to feet and kips
XX, YY, UUU, FF	Work areas for determining the stresses in the elements

8. Subroutine STNON

Variable Name	Description
IE	Index for two end springs of a beam
NAK	Total size of AK
P	Percentage of full static load

9. Subroutine FORGEN (To be included later -- in final report)

10. Subroutine SOLVE

Variable Name	Description
HH, DT1, DT2	Parameters in Newmark Beta method

11. Subroutines BEAM2

Variable Name	Description
MS3, MS6, POSTS, PRES	Temporary work spaces for matrix multiplications (BE
SI, CO	Sine and cosine
XX	x coordinates of the nodes
YY	y coordinates of the nodes
BE	Modulus of elasticity
BA	Area of cross section
BI	Moment of inertia (only in BEAM2)
BW	Unit weight

12. Subroutine BMK

Variable Name	Description
KYA, KYB, R,C, SBM ASC, KYAC, SAC, BS, KYB1, KYB2, SB,	Local names for arguments, stiffness

13. Subroutine BAR

Variable Name	Description
XX, YY, BE, BA, BW	Same as in BEAM2

14. Subroutine CST

Variable name	Description
XX	x coordinates of the nodes
YY	y coordinates of the nodes
THK	Thickness of the element
TE	Modulus of elasticity
TPR	Poisson's ratio
TW	Unit weight
B,C	Area coordinates

15. Subroutine BSUB

Variable name	Description
A, NROW, M, NA	Same as in subroutine DECOMP

16. Subroutine MODBM

Variable Name	Description
DHR	Increment in hinge rotation
AO, AOC, A1	Current axial and bending stiffnesses
KCHFR	Indicator of rupture

17. Subroutine HYST

Variable Name	Descriptions
DXN, DXP	Differences from negative and positive yield points
ENDI, ENXPS	Dissipated and permanent set energies
CXYN, CXYP	Current positive and negative yield points
FRP, FRN	Positive and negative rupture forces
KY	Current stiffness key
NEWR	Counter on rupture
WEWY	Counter on yield
PDROP	Loss of force due to rupture
S, SS, SY	Stiffness parameters
X	Displacement
XPS	Permanent set displacement
XRP, XRN	Positive and negative rupture displacments
XYP, XYN	Positive and negative yield displacements

17. Subroutine MULT

Variable Name	Description
A	Matrix
X	Vector to be multiplied with A
Y	Resultant vector (Y) = [A] (x)
NROW	Number of rows in A
M	Half band width (including diagonal) of A

DISTRIBUTION LIST

DDC Cameron Station Alexandria VA 22314	2
AUL (AUL/LSE-70-239) Maxwell AFB AL 36112	1
ASD/ENFEA Wright-Patterson AFB OH 45433	1
USAF/SAMI Washington DC 20330	1
TAWC/TRADOCLO Eglin AFB FL 32542	1
AFATL/DL Eglin AFB FL 32542	1
AFATL/DLOSL Eglin AFB FL 32542	2
ASD/ENYEHM Wright-Patterson AFB OH 45433	1
Ogden ALC/MMNOP Hill AFB UT 84406	2
AFWL/LR Kirtland AFB NM 87117	2
AFIS/INTA Washington DC 20330	1
AFATL/DLYV Eglin AFB FL 32542	5
Texas Tech University Civil Engineering Department Lubbock, Texas	10